Carlson Takeoff R2

Carlson Software Inc.

User’s manual

August 13, 2007

Contents

Introduction 1

Installing Carlson

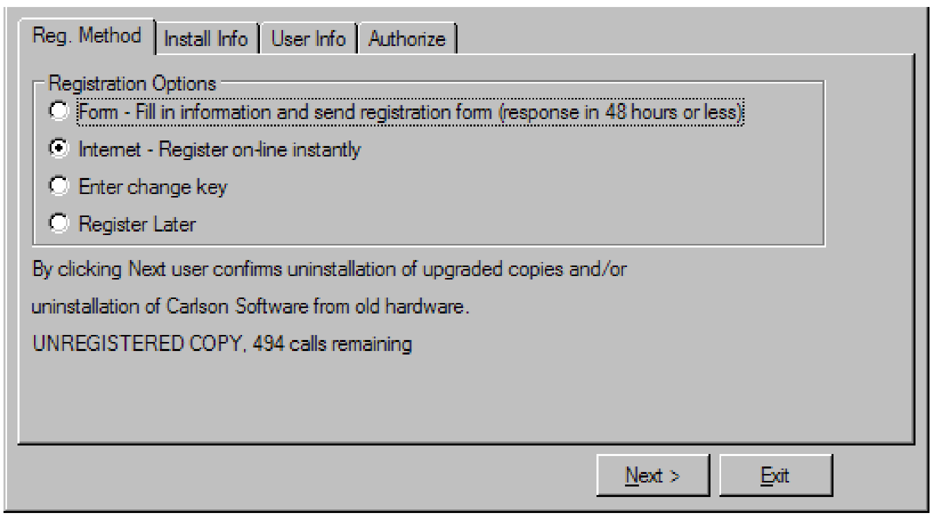
Please refer to the installation guide included in the Carlson package for the latest installation instructions. To complete the Carlson installation, start Carlson and complete the registration process as explained in the following sections.

For more installation options in the future, refer to the Carlson Software download page on our website at http://www.carlsonsw.com.

Carlson Registration

Each Carlson program is licensed for use on **one** workstation which must be registered. The registration records your company name and AutoCAD serial number. To register your copy of Carlson, start Carlson and choose "Register Now". The following dialog will appear.

Note: Carlson Software will no longer issue change keys over the telephone. There are four registration options.



Fax: This method allows you to print out the required information on a form which you then fax to Carlson Software. The fax number is printed on the form. The change key will be faxed back to you within 72 hours.

Internet: Register automatically over the Internet. Your information is sent to a Carlson Software server, validated and returned in just a few seconds. If you are using a dial-up connection, please establish this connection before attempting to register.

Enter pre-authorized change key: If you originally chose the Fax method above, you will need to choose this method now to enter the change key that is faxed back to you.

Register Later: If you wish, you may defer registration up to 30 days. After this time, Carlson will enter demo mode which displays a message each time a Carlson command is run.

After you select the registration method, choose Next and select the type of installation you are performing, choose Next again to review the copyright information and to fill out the required information. At this point, if you are using the Fax method, press the Print Fax Sheet button. If you are registering using the Internet method, press Next and the process will start.

If you have any problems with Internet registration, please repeat this process and use the Fax method. The registration form is available on the Carlson Software website at http://www.carlsonsw.com/registration.html.

Tip: If Carlson is running, you may access the registration dialog by choosing *About Carlson* from the Help menu, then pick the Change Registration button.

Obtaining Technical Support

via Discussion Groups

- Carlson Software operates user discussion groups located at news://news.carlsonsw.com. You can participate in user-to-user discussions on tips, tricks and problems. Our staff monitors these groups to ensure that all the issues are addressed. Visit our website at http://www.carlsonsw.com for information on how to access these groups.
- You may also access the Carlson Software Knowledge Base. Visit it directly at http://update.carlsonsw.com/kbase_main.php.

via Electronic Mail

- The Technical Support e-mail address is support@carlsonsw.com.

via Phone/Fax

- Phone: (606) 564-5028
- Fax: (606) 564-6422

via Web Site

Check the Carlson Software web site at http://www.carlsonsw.com for:

- Knowledge Base, discussion groups, technical support documents and newsletters
- Carlson Software manuals (PDF) and training movies
- Training and seminar schedules
- Step by step procedures on popular called-in topics
- Carlson Software and Autodesk downloads and updates (Feel free to register for automatic update notification of updates when you come to that area.)

via Training

- Basic, advanced and update training is available from Carlson College. Enroll on our webpage or call 606-564-5028 and ask for Carlson College.

Setting Up a Project

Over 200 program settings can be specified in the Configure command under the Settings menu. These values are used to initialize Carlson program options when opening a new or existing drawing. Among these settings is the coordinate point number format, file and printer output options and settings for each module.

To set the AutoCAD drawing defaults, edit the template drawing (.DWT file). This drawing is loaded when new drawings are created. In the template drawing you can set the layers and AutoCAD variables. For example you could create your standard layers and set variables as you like such as BLIPMODE off. For Carlson, the drawing template should be set to Carlson##.dwt where the ## is the AutoCAD version number. For Carlson running in AutoCAD 2007, the template name is Carlson07.dwt. The Carlson template is located in the Carlson

support directory (i.e. C:\Carlson2008\SUP\Carlson07.dwt). To customize the template, run the AutoCAD *OPEN* command and choose the drawing template. In the Select File dialog, set the type of file to Drawing Template (DWT) instead of regular drawings (DWG). Then make your changes and *SAVE* the drawing as Carlson##.dwt.

When starting a new drawing, one of the first steps is to run Drawing Setup under the Settings menu. Drawing Setup sets the drawing scale, the unit mode as either English or metric, and the text, symbol and linetype size scalers. The initial values for these Drawing Setup variables are set in Configure. When a drawing is saved, the Drawing Setup variables are saved with the drawing.

In Carlson, the AutoCAD text style height should be set to zero. The Carlson routines will set the text height according to the drawing scale and text size scaler set in Drawing Setup. For example, if the horizontal scale is set to 50 and the text size scaler is 0.1, Carlson will draw the text with a height of 5 ($50 * 0.1$). Then when the drawing is plotted at 1"=50', the text will be 0.1 inches. Use the AutoCAD *STYLE* command to set the text style height to zero.

The Set Data Directory command in the Settings menu can be used to specify the directory for the project data files. By default the drawing is stored in the Carlson WORK directory and the data files are stored in the DATA directory. The drawing file is the AutoCAD (.DWG) file. The data files are the coordinate (.CRD) file, profile (.PRO) file, grid (.GRD) file and other Carlson data files. In Configure>Project/Data Folders, there is an option to store all data files in the directory of the drawing. With this option active all the files for the drawing C:\Carlson2008\Work\JOB500\JOB500.dwg would be stored in C:\SCAD2006\WORK\JOB500.

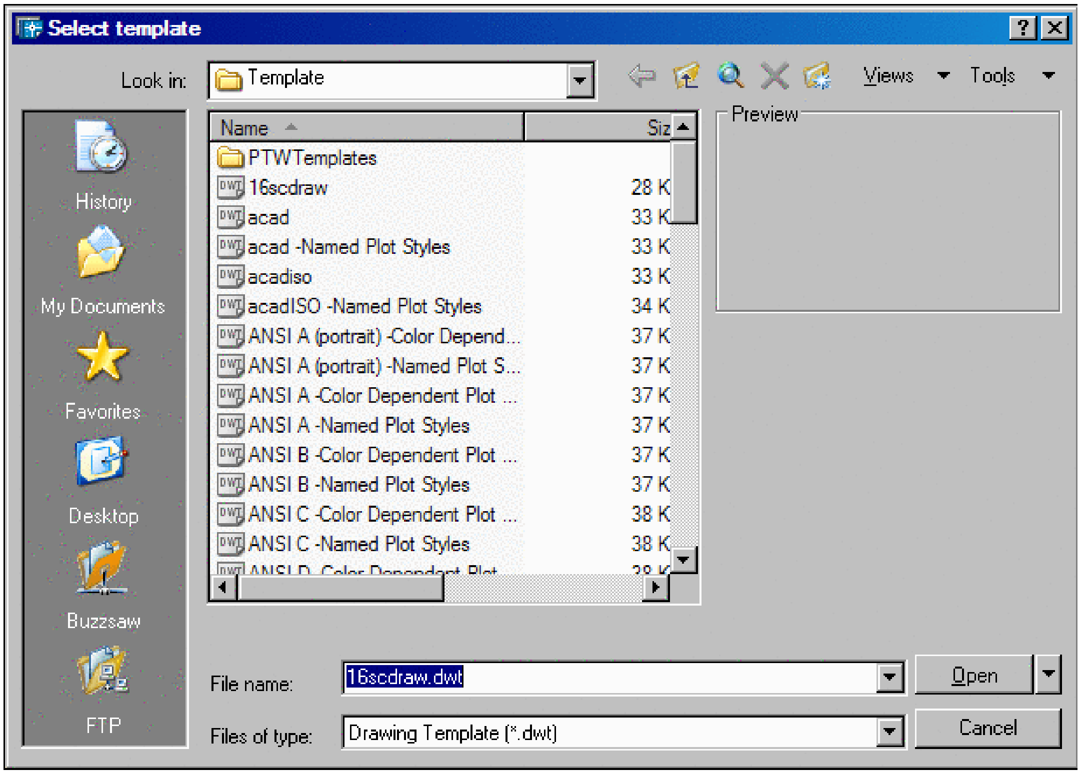
Another level of file management is the automatic project file recall. Every drawing remembers the data files that are being used for the drawing. When the drawing (.DWG) file is saved with the SAVE, SAVEAS, or QSAVE command, Carlson writes a settings file that contains all the active data file names. Then when the drawing is reopened, the data files default to their previous settings. For example, you won't have to choose which coordinate file to use unless you want to change it. The settings file is stored in the same directory as the drawing file and has the same name as the drawing with a .INI extension. For example, a drawing survey.dwg would have a settings file called survey.ini. You can turn off the INI files with the Save Drawing INI Files option in Configure under General Settings.

New/Startup Wizard

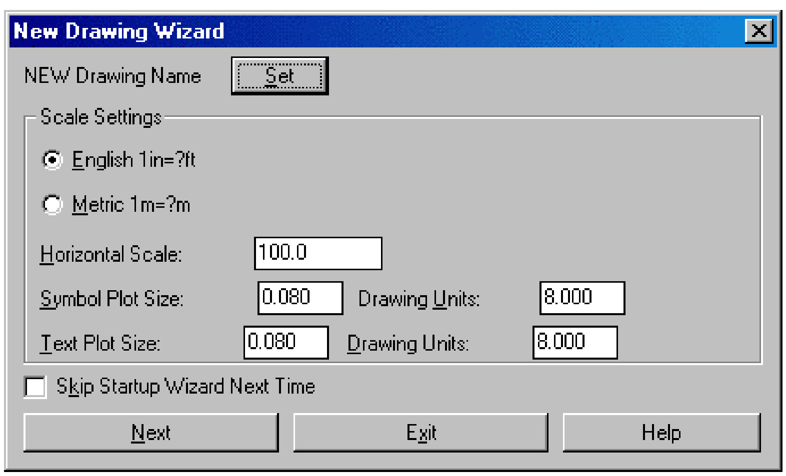
The AutoCAD New command is used for starting a new Carlson drawing. This page describe this AutoCAD New command and the Startup Wizard, along with the Carlson variables, associated with it.

Built into this routine is a Startup Wizard that can step you through and make the new Carlson drawing setup process easier. For creating a new drawing in Carlson, the Startup Wizard guides you through starting and setting up the drawing. This wizard is optional, and can be turned on or off in the Configure > General Settings command, which is part of the File pulldown. There is also a dialog box option, shown and mentioned below, that allows you to disable this feature. You can also exit out of the Startup Wizard at any time.

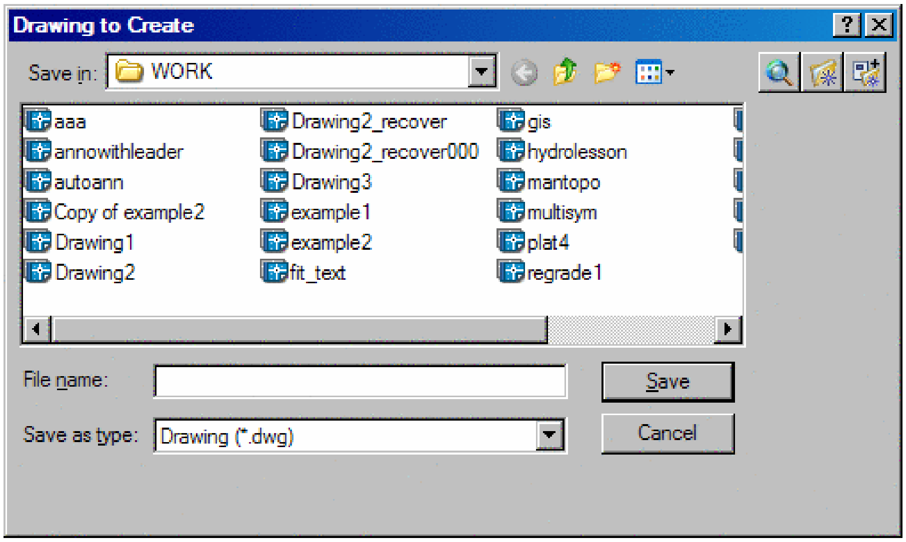
When the AutoCAD *New* drawing command is executed, you first get the standard AutoCAD Select template dialog box. While there are many templates to choose from, and there is an Open option, typically you want to go with choosing the Carlson drawing template (CARLSON07.DWT). The drawing template will set of some basic drawing parameters such as default layer names.



After selecting the template, the Carlson Startup Wizard begins by opening the *New Drawing Wizard* dialog box.

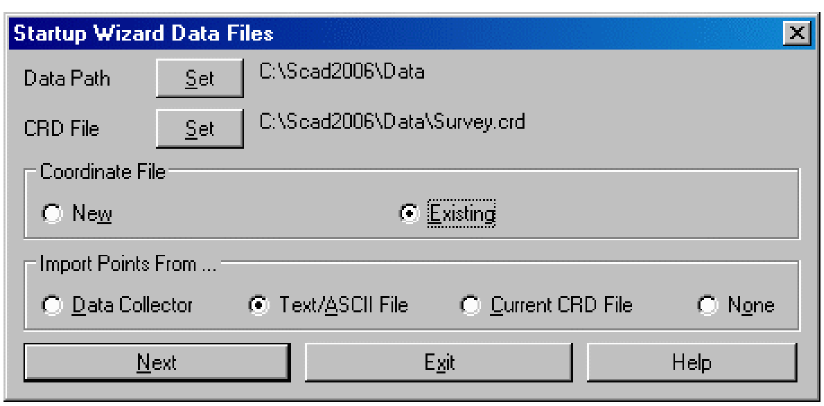


This dialog is used to set the drawing name and scale. The first step to do is set the drawing (.DWG) name by picking the *Set* button. This brings up the file selection dialog. Change to the directory/folder ("*Save in*" field) where you want to store the drawing. You can either select an existing folder or create a new folder. To select an existing folder, pull down the *Save in* field to select a folder or drive, click the Move Up icon next to the *Save in* field and/or the pick the folder name from the list. To create a new folder, pick the Create New Folder icon to the right of the Save in field. Then type in the drawing name in the *File name* field and click the *Save* button.



After setting the drawing name, you can set the drawing horizontal scale, symbol size, text size and unit mode (English or Metric). Notice that at the lower left corner of the *New Drawing Wizard* dialog there is an option to Skip Startup Wizard Next Time. Typically, you would leave this option unchecked, as the Wizard is a handy tool for new drawing setup. Now click the Next button.

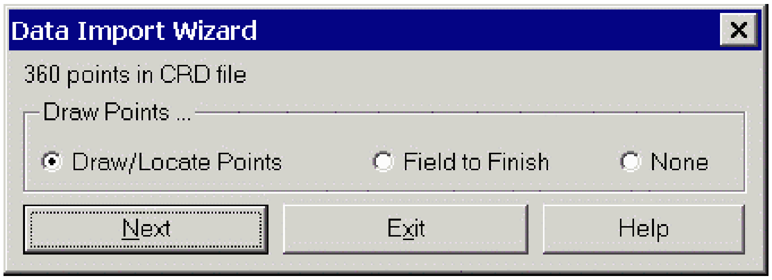
The next startup dialog sets the Data Path and CRD File. The Data Path is the folder where Carlson will store the data files such as raw (.RW5) files and profile (.PRO) files. The *Set* button for the Data Path allows you to select an existing folder or create a new folder. See the Set Data Directory command for more information. The coordinate (.CRD) File is the coordinate file for storing the point data. There is an option to create a new or existing coordinate (.CRD) file. The new option will erase any point data that is found in the specified CRD file. The existing option will retain any point data in the specified coordinate (.CRD) file. If the specified coordinate (.CRD) file does not exist, the wizard will create a new file.



The next wizard step depends on the Import Points option. The Data Collector option will start the data collection routines to download data from a collector. The Text/ASCII option will import point data from a text/ASCII file. See the Data Collection and Import Text/ASCII File commands for more information on running these routines. The Current CRD File option is a popular one to choose for bringing in coordinates. If the None option is set, then the Startup Wizard is finished.

Once point data has been imported from the data collector, text/ASCII file or CRD file, the wizard guides you through drawing the points. There are options to run Draw/Locate Points, Field To Finish or None. If None is

selected, then the Startup Wizard is finished. Draw/Locate Points will import the points into the drawing using the same symbol and layer for all the points. From the Draw/Locate Points dialog, set the symbol, layer and point attributes to draw (description, elevation) and then pick the Draw All button. The Field To Finish command will import the points into the drawing using different layers and symbols depending on the point descriptions that refer to the code table defined in Field to Finish. Also Field to Finish can draw linework. See the Draw/Locate Point and Field To Finish commands for more information on running these routines. After drawing the points, the wizard will zoom the display around the points. Then the wizard is finished.



Pulldown Menu Location: File
Keyboard Command: new (AutoCAD)
Prerequisite: None

Command Entry

Commands may be issued by selecting a pulldown menu, screen menu, digitizer tablet item, or by typing a command at the AutoCAD command prompt. Pressing Enter at the command prompt repeats that last command. Pulldown menus have a row of header names across the top of the screen. Selecting one of these header names displays the possible commands under that name. Screen menu items are shown in the screen menu (typically on the right side of the screen). The screen menu can be toggled off and on inside of the AutoCAD Preferences dialog. The Pulldown menus are the primary method for Carlson command selection. Each section of this manual shows the pulldown menu which contains the commands that are explained in that section. Pulldown menus are sometimes also referred to as dropdown menus.

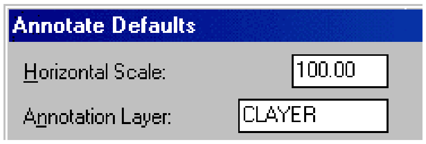
Command availability depends on which menu is loaded. Carlson menus have a mixture of both Carlson and AutoCAD commands. This allows you to execute the commonly used AutoCAD commands from the menus while running Carlson.

Quick Keys are user-defined short cut names that can be typed in to start commands. To review the current set of Quick Keys, run the Quick Keys command in the Settings pulldown menu. Quick Keys are explained in more detail in the next section.

Layer and Style Defaults

Many Carlson commands have default layers such as AREATXT for area labels and BRGTXT for bearing and distance annotations. These layers can be specified in dialogs for the corresponding commands and several can be set in Configure. Sometimes you may want to use the current layer and it can be an extra step to have to open the dialog to set the layer. In this case, instead of using the default layer that set in the dialog, the default layer can be set as "CLAYER" which will use the current layer. For example, if the annotation layer is set to CLAYER then annotation will be drawn in the current layer instead of BRGTXT or whatever the annotation layer used to be.

This same concept applies for text styles. Several commands have specific text styles and if you want to use the current style instead of the command style, use the name "CSTYLE" for the style name.



Carlson File Types

.AAN Auto-Annotate Settings
.ADF Annotation Default Settings
.ARX AutoCAD Runtime Extension For Carlson Program
.ATR Strata attribute definitions
.AVG Mining Composite Quality Analysis
.BLK Mining Block Model
.CAL Mining equipment calendar
.CAP Capacity file for hydrology (stage-storage)
.CDF Geology Channel Sample File Format
.CDS MDL Laser Raw Data
.CDT Mining custom date table
.CFG Configure Configuration Settings
.CFZ Cut/Fill Color Map Zones
.CGC C&G Coordinate File
.CGR C&G Raw Data
.CH Corehole definition
.CL Centerline file
.CLT Culvert Settings
.CN Hydrology CN Factors
.COG Cadvantage Coordinate Data
.COT Multiple Outlet Design Data
.CQT Mining custom quantity table
.CRB Template Curb Definition
.CRD Coordinate file (point#, northing, easting, elevation, description) in binary form
.CTL SDMS Format Raw Data
.CTR Auto-Run Strata Isopach Maps
.CUI Customized User Interface AutoCAD Menu
.CUT SMI Format Cutsheet
.CVT SEDCAD Format Hydro Network
.DAT GPS Localization Definition
.DCF Deed Correlation File
.DCL Carlson dialogs
.DEM Digital Elevation Model
.DEQ Drillhole equations
.DHF Drillhole Text File
.DHT Dragline History
.DIL StrataCalc Convert As-Determined Qualities
.DLL Carlson programs files
.DTF Drillhole Data Format

.DTS Drillhole Text Settings
.DWG AutoCAD drawings
.DXF Drawing Exchange Format
.DZR Dozer Push Settings
.EQO Mining equipment options
.EQU Mining equipment definitions
.ERD Erodible Channel Settings
.EW Earthwork Section End Areas
.EXE Carlson programs
.FCL Feature Code List for SurvCE
.FEN Fence Diagram Settings
.FLD Field to Finish code definitions
.FLT Triangulation mesh
.FMS Report Formatter Settings
.FRM Ferm Codes
.GCL Geologic Column Settings
.GEO Settings for Draw Geologic
.GFU Grid File Utilities macro command recorder file
.GIS GIS prompting definitions for note files
.GPF Grade Parameter File For Block Model
.GRD Grid file - a DTM surface model
.GRR Lot Edge Grading Rules
.GSQ Grid sequence history file for mining
.H1 Hydrograph
.H2I HEC-2 Water Surface Profile
.HYD Watershed settings
.HZN Horizon code definitions
.IMP Drillhole Import Format
.INI Program user preferences settings
.LAY Layer State
.LEV Level Raw Data
.LGD Annotate Draw Legend definition file
.LOT Lot file for Survey
.LPT Coordinate Transformation Point File
.LQE Least-Squares Measurement Error Settings
.LSP Carlson Lisp programs
.LSQ Least-squares input data
.LSS Localization Settings
.LTD Lot Attribute Definition
.LTN Lot Network Settings
.MAT Materials Library
.MDN Template Median Definition
.MIN Underground Mining Report
.MNC Compiled menus
.MNL Carlson Lisp programs for corresponding menu startup
.MNR Compiled menus
.MNS Compiled menus
.MNU Carlson source menus
.MNX Compiled menus
.MPD Mining Timing Project File

.MPJ Mining project definitions
.MXS Cross section alignment
.NOT Note file - additional descriptions for points in corresponding .CRD file
.OWN Mining Ownership Data
.PAN Mining Underground Panel Data
.PAR Parameters for mining quality compliance
.PAT Hatch Pattern Definitions
.PDD Enter Deed Description data file
.PFF Mining Pit Format File
.PFS Draw Profile Settings
.PHN Modem Settings For Carlson Field
.PIT Mining Pit Parameter File
.PLN Plan view polyline file for Dozer 2000
.PNA Mining panel attributes
.PPQ Polygon Processor Query
.PRE Pre-calculated grids for strata model
.PRF Point Range File
.PRJ Project settings file
.PRO Profile (station, elevation, descriptions)
.PSZ Pipe Size Definitions
.PTA Mining pit attributes
.PVF System Variable Settings
.PVL Sewer Network Plan View Label Settings
.PVS Mining Projection and Ventilation Settings
.RCF Drainage Runoff Library
.RCL Runoff Coefficient Layer Definitions
.RDF Road Design File
.RDN Road Network File
.RDS Roadside Ditch File
.REC Seismic Record Output
.REP Hec-Ras Report File
.RES Mining Auto-Run Residuals Settings
.RME Reame Slope Stability Settings
.RMP RiverMorph Project
.RNF Road Network Stakeout for SurvCE
.ROW Right-of-Way offsets for Lot Network
.RPT Report Formatter Data
.RSV Mining Reserve Description
.RUN Auto-Run Strata Grids definition file
.RW5 Raw file of traverse data
.SC Coal Section Configuration
.SC5 SurvCE Raw Data Archive
.SCR COGO script file
.SCT Cross section data (station, offset, elevation, descriptions)
.SDF Strata Definitions
.SDT SB-Slope Stability File
.SEQ Dragline sequence file
.SEW Sewer network file for hydrology
.SGD Slope Group Definition for Design Pad Template
.SIF Surface Inspector File

.SLB Slide library
.SLD Slide image
.SLI Symbol Library
.SST Draw Section Sheet Settings
.STG Stage-discharge data for hydrology
.STR Sewer Structure Library
.SUP Superelevation file
.TAB Hydrology Peak Flow Data
.TCF Digitizer Tablet Configuration File
.TCH Trench Structure Template
.TIM Mining Timing Settings
.TIN Triangulation Surface
.TKD Takeoff Material Definitions
.TOP Topsoil Definition for Road Design
.TPC Template point centerline definition
.TPL Template file for Section-Profile
.TPP Template point profile definition
.TPT Template transition file for Civil Design
.TRG Takeoff Layer Target Settings
.TRI Triangulate & Contour Settings
.TSF Road Template Series File
.UAO Report Formatter Attribute Options
.UAT Report Formatter User Attributes
.UDP User-Defined Projection For Grid Coordinates
.VOL Auto-Run Strata Quantities definition file
.WST Watershed Structure File
.XML LandXML Data Exchange File
.ZIP Compressed Data Files
.ZON Earthwork Zone Definitions

Standard Report Viewer

Many Carlson routines display output in the Standard Report Viewer as shown below. The report can be edited directly in the report viewer. Report Viewer commands are described below.

Open: This allows you to open an ASCII file and display the contents in the report viewer.

Save: Save the contents of the report viewer to a text file.

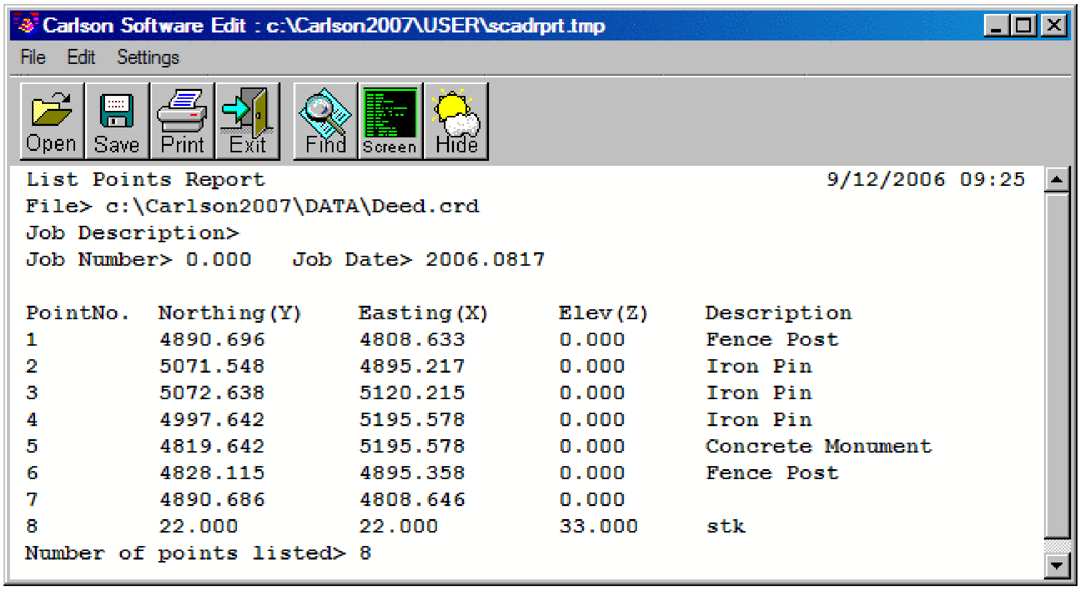
SaveAs: This allows you to save the contents of the report viewer to a file.

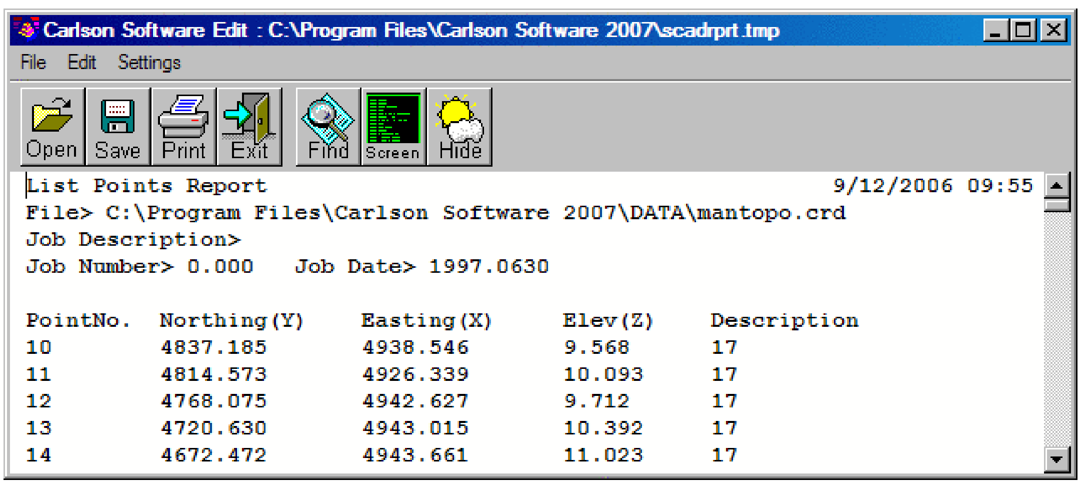
Append To: This allows you to append the contents of the report viewer to another file.

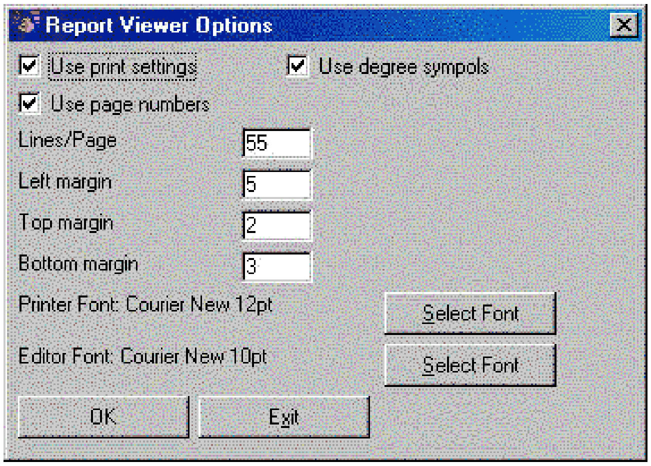
Print: Print the contents of the report viewer. This will open the standard windows Print dialog where you can choose the printer and modify any of the printer settings before you actually print.

Screen: Draws the report in the current drawing. The program will prompt you for a starting point, text height, rotation, layer and whether you want it inserted as Mtext or Text.

Undo: Reverses the effect of your last action. If you mistakenly deleted some text, stop and choose the Undo command to restore it. The key combination Ctrl+Z also performs this action.







Select All: Selects all the text in the report viewer.

Cut: Deletes the selected text and places it on the Windows® clipboard.

Copy: Copies the selected text to the Windows® clipboard.

Paste: Inserts ASCII text from the Windows® Clipboard into the report viewer at the cursor.

Search: Opens the Find Text dialog. Allows you to search for text in the report viewer.

Replace: Opens the Find and Replace Text dialog. Allows you to search for text and replace it.

Options: Opens the Report Viewer Options dialog. In this dialog, you can specify print settings, such as lines per page and margins. You can also specify the font used in the report viewer. This font is used for both the display and for printing.

Hide: This button allows you to minimize the report viewer window and give focus back to the Carlson CAD screen. This allows you to return to working on the Carlson CAD screen without closing the report. You can re-activate the report by picking on the minimized report viewer icon.

Report Formatter Dialog

A number of Carlson routines use a dialog box called Report Formatter Options to allow you to specify how and which results of calculations should be presented in the report. This report routine lets you select a set of data to report and the format of the selected data. The report can be displayed in either the standard report viewer as described in the previous section, Microsoft® Excel or Microsoft® Access.

The data set in the Report Formatter may be thought of as a spreadsheet, where columns are various fields related to a single item such as time period, drillhole, area and etc. Each new row represents a new item. Descriptions of these field names are displayed in the Available list of the Report Formatter. To include a data field in the report, highlight the field name in the Available list on the left and pick the **Add** button. This moves the field name to the Used list on the right. The order of items in the right list defines the order in which they will be displayed. To change the order of a field, highlight the field name in the Used list and pick the Up/Down arrows button on the left side of the Used list. In addition, the items may be sorted as specified by the user in the right column. For each field, you can set the sort method as Hold, Up, Down or Ignore. Fields are first sorted by the first column, then fields with the same value in the first column are sorted as specified for the second column, and so on. These subsequent sortings do not modify sortings of previous columns. If you specify **Hold** sorting for some column (even the first one) then no sorting will happen in subsequent columns either. For example, you may want to sort production by mining panel name but not by month. The **Ignore** sorting option let's you skip sorting for a column and then still

apply sorting for the next columns.

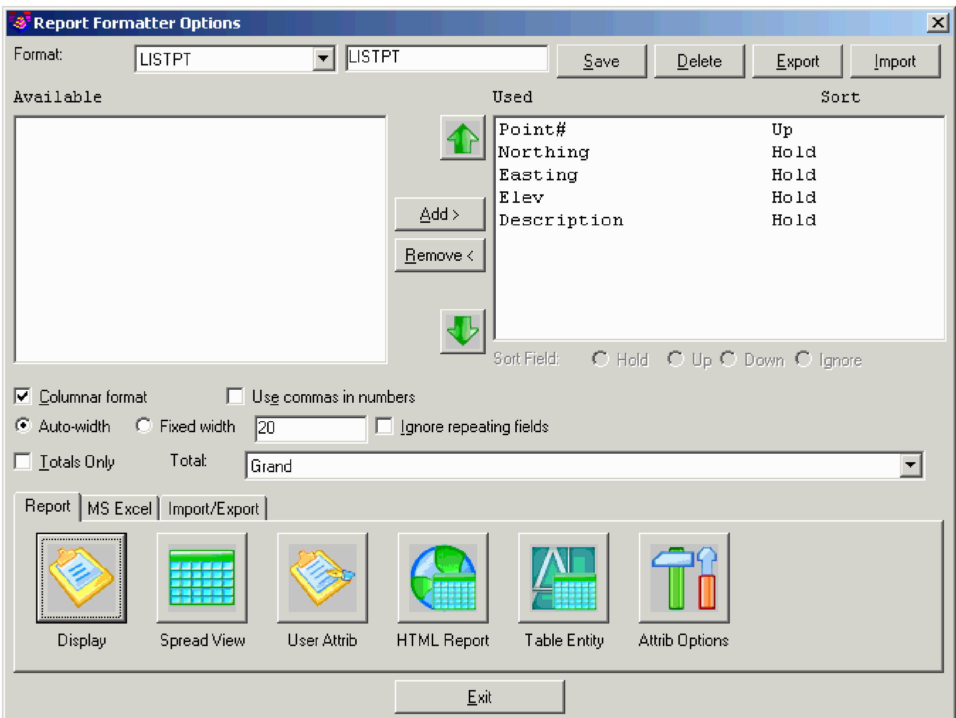
The **Columar format** makes columns for each field with rows for each record. Otherwise report data is output in a single column.

Use commas in numbers will put commas in number fields for every three digits.

The width of the field columns can be either **Auto-width** sized to fit the longest value in the column or at a specified **Fixed width**.

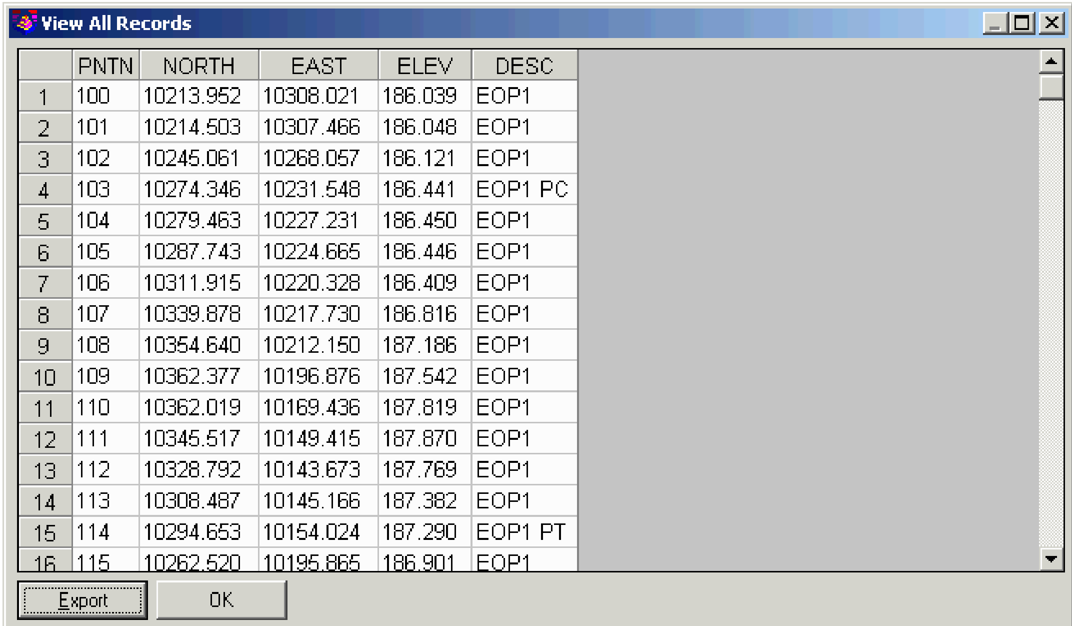
The **Ignore repeating fields** option will report a field record only once at the top row when the same value is repeated on the following rows.

The **Totals Only** option will report just the totals for each field. The **Total** pull-down list groups the report by the selected fields.



To generate the report after selecting columns and other preferences, click on **Display** button. It will bring up a standard report viewer showing the report data. Upon exiting the viewer, you come back into the Report Formatter for further data manipulation if needed.

The **Spread View** button shows the report data in a spreadsheet for review and edits. There's also an Export button with the same output options as the Export function under the Import/Export tab.

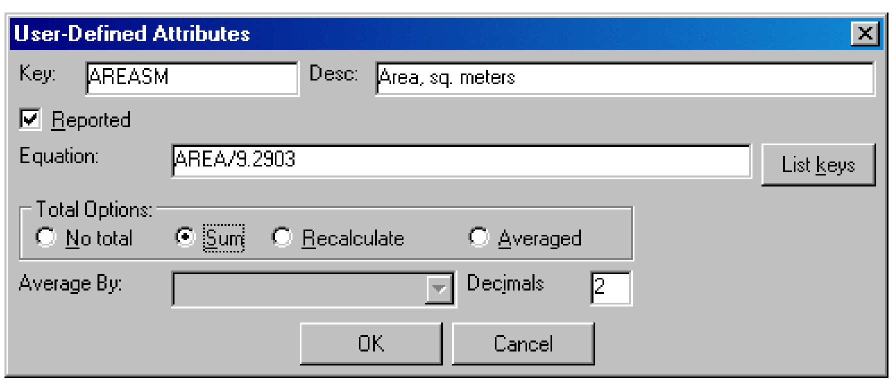


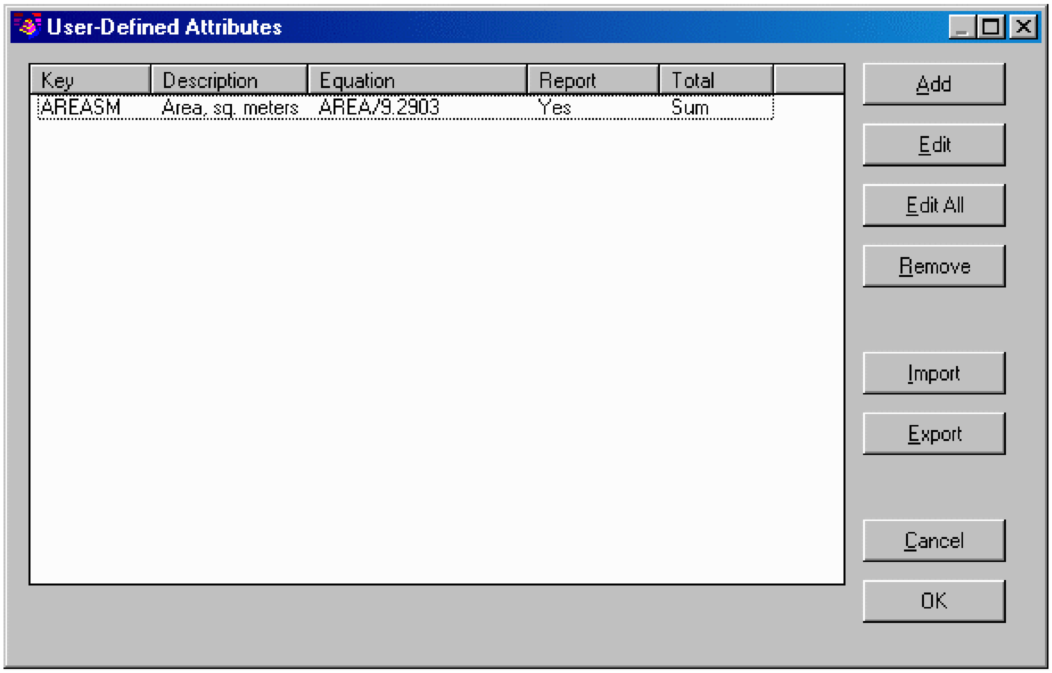
	PNTN	NORTH	EAST	ELEV	DESC
1	100	10213.952	10308.021	186.039	EOP1
2	101	10214.503	10307.466	186.048	EOP1
3	102	10245.061	10268.057	186.121	EOP1
4	103	10274.346	10231.548	186.441	EOP1 PC
5	104	10279.463	10227.231	186.450	EOP1
6	105	10287.743	10224.665	186.446	EOP1
7	106	10311.915	10220.328	186.409	EOP1
8	107	10339.878	10217.730	186.816	EOP1
9	108	10354.640	10212.150	187.186	EOP1
10	109	10362.377	10196.876	187.542	EOP1
11	110	10362.019	10169.436	187.819	EOP1
12	111	10345.517	10149.415	187.870	EOP1
13	112	10328.792	10143.673	187.769	EOP1
14	113	10308.487	10145.166	187.382	EOP1
15	114	10294.653	10154.024	187.290	EOP1 PT
16	115	10262.520	10195.865	186.901	EOP1

You may define new fields as equations based on existing fields. Click on the **User Attributes** button to add a new field name. A list of the existing attributes is available for reference.

User attributes may have one of the several summation options just like program-generated ones (except that for them these options are set by program). The summation level is defined by the "Total" pop-up list in the middle of the dialog. By default only grand total will be displayed at the bottom of the list. By picking the next item in that box, you will get subtotals added each time the value in first column is changed. It makes most sense to use this kind to summation if the corresponding column is sorted. For example if the first column is "Area Name" and it is sorted, and "Total" is set to "Grand, Area Name" the report will have a sub-total for each distinct Area Name. This feature makes the Report Formatter a very flexible tool for results exploration, even before or without using a spreadsheet.

Various forms of reports may be saved and recalled using controls in the top line of the dialog.

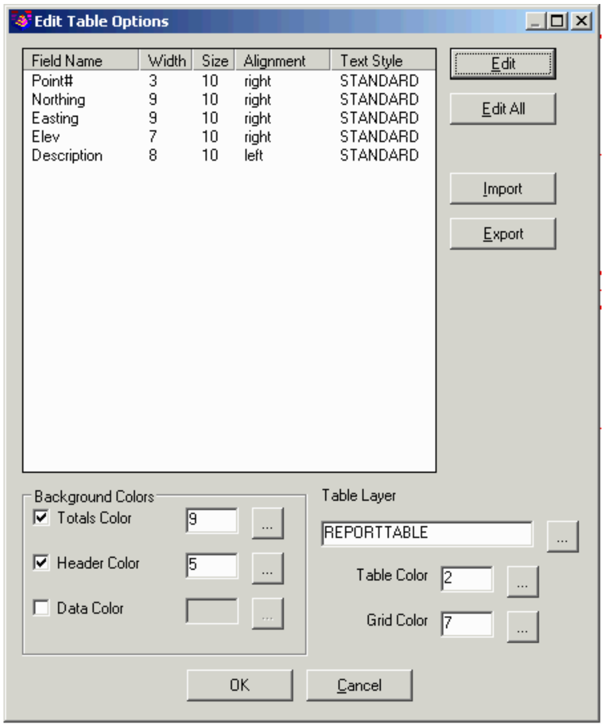




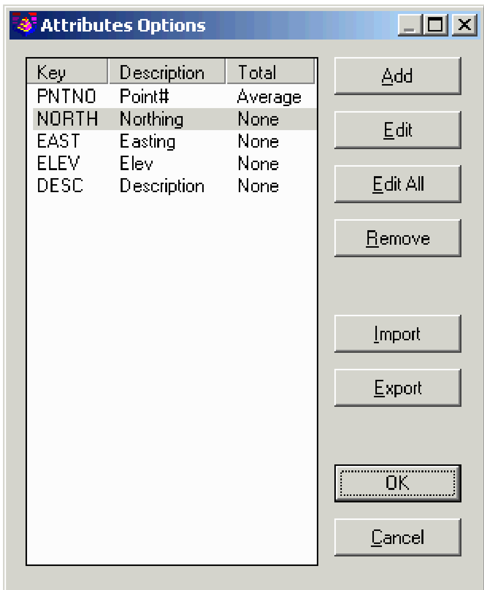
To save a new version of the format, type in a new name (or use default to overwrite old one) and click on the Save button. The next time that you come to the Report Formatter from the same Carlson routine it will recall this last format. To pick another format just pull down on list of formats in the left top corner and pick which format to use. To Delete an unwanted format, pick it from the list first and then click on Delete button.

The **HTML Report** button generates a report in html format and display the report using the html viewer that is configured on your computer such as Microsoft Internet Explorer.

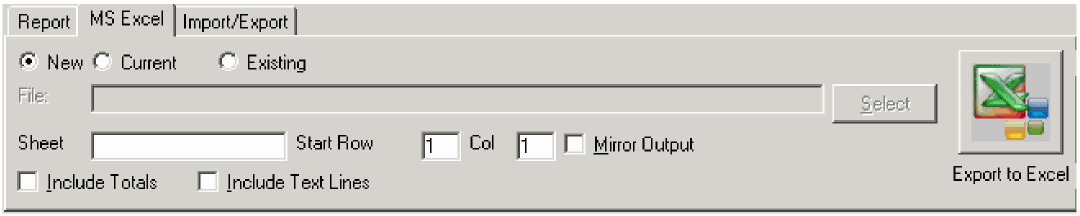
The **Table Entity** button creates an AutoCAD table entity in the drawing with the report data. When this button is picked, the data for the table is put into a queue and the table is not drawn until the Report Formatter is closed. Then the program prompts for a location to draw the table and there's a Table Options dialog to set the parameters for the table including the header names, sizes, alignments, styles, colors and layers.



The **Attribute Options** allow you to control several parameters of each field including title names, number of decimal places and totaling method by sum, simple average, weighted average or none.



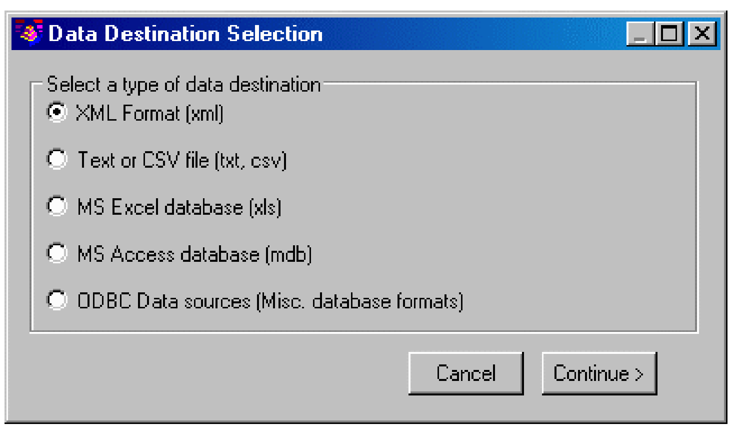
Under the **MS Excel** tab, there are several Microsoft® Excel export options provided. You may specify a spreadsheet file to load before export, as well as a left upper cell to start with and sheet name to use. The Mirror Output option switches the report rows into columns and the columns into rows. Totals and text lines which are reported when using built-in viewer may be skipped when using Microsoft® Excel export.



Under the **Import/Export** tab, the **Export** button has several output options for the report data. These other data output options include XML format files, comma-delimited text or CSV file, direct export to Microsoft® Excel and Access and to other database formats as well.

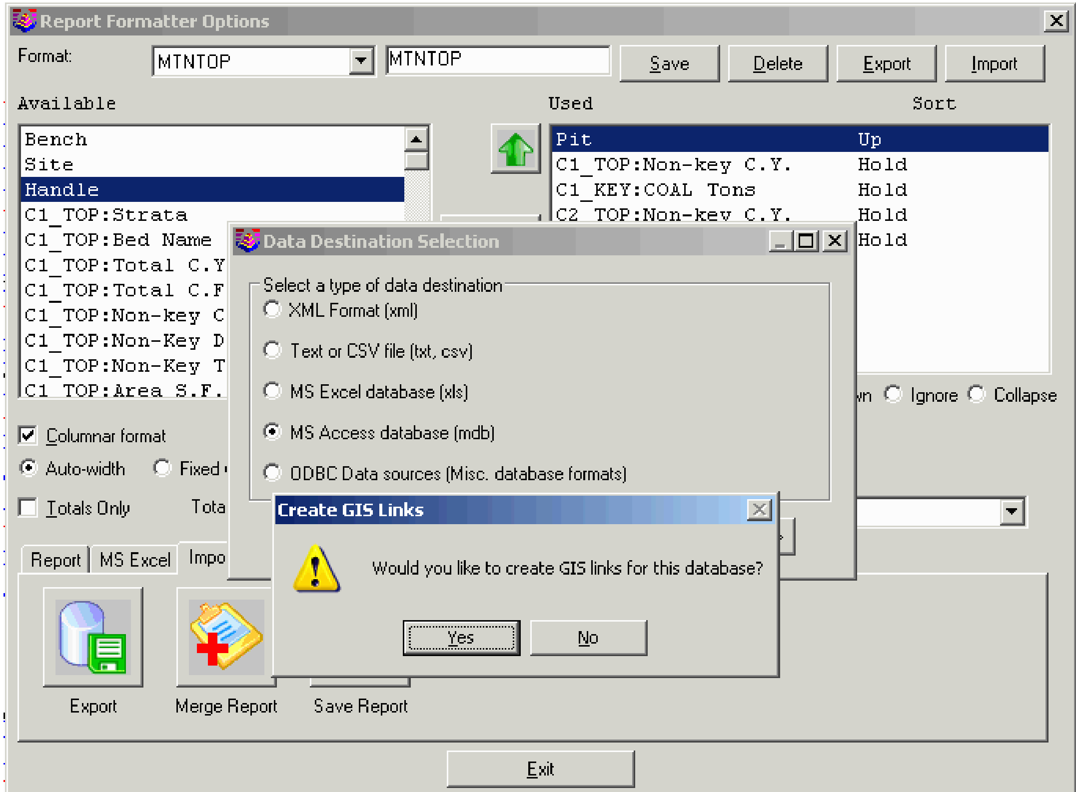
Using the **Merge Report** function, data can also be merged to combine current and old reports.

The **Save Report** function saves all the report data values as well as all the report format settings into a single .rpt file that can be shared with others, merged with other reports or loaded at a later time.



For commands that process reports using perimeter polylines, the Report Formatter has an option to create GIS links between the polylines and the database records when the Export to MS Access function is used. Some commands that apply are Surface Mine Reserves with the pit polylines and Underground Timing with the panel polylines. When the polyline data is available for the GIS Links, there will be a report field called Handle. This Handle field is the AutoCAD entity name for the polyline and serves as the hook for the GIS link. The Handle field does not need to be put into the report Used list in order to create the links. When the Export function is called with the MS Access method, there is a pop-up window prompt for whether to create the GIS links. When

these links are created, you can then use the GIS Data commands from the GIS module to manage and report the data.



Instruction Manual and Program Conventions

Westwood
Italic text represent responses by the user that should be typed in and followed by the Enter key.

Number/<Pick point>:
Bold text represents prompts or questions that the computer program will ask the user.

<**90.0000**>
Values enclosed in corner brackets represent default values obtained by pressing Enter with a blank response to AutoCAD or Carlson commands.

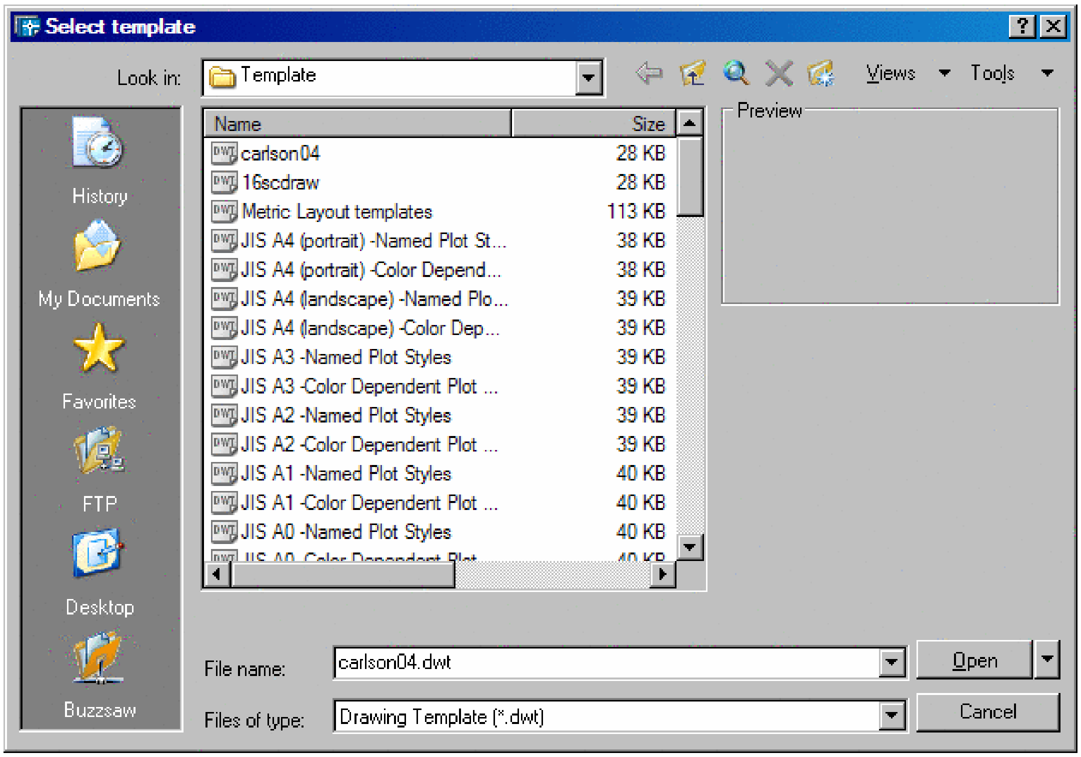
[end on]
Lower case text enclosed in brackets in Command prompts denotes an *OSNAP* mode that is turned on by the command.

File Menu 2

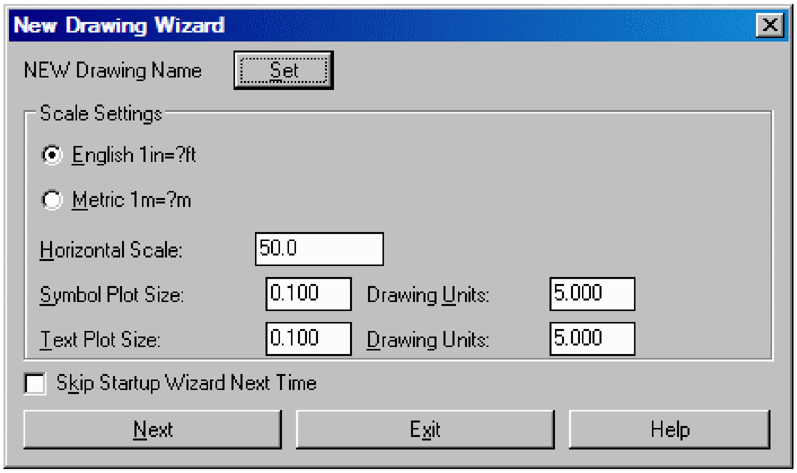
New

This command allows you to create a new drawing file. This routine defines the settings for a new drawing. You can start a new drawing file by selecting New, and then picking a template file. The first dialog for the New command, called Select Template, lists all template files that currently exist in the drawing template file location. Choose a file to use as a starting point for your new drawing. A preview image of the selected file is displayed to the right.

The template file (.DWT) that you use will depend on the version of AutoCAD that you are running. For AutoCAD 2000-2002, the Carlson template file is carlson02.dwt. For AutoCAD 2004, it is carlson04.dwt. For AutoCAD 2005, it is carlson05.dwt. For AutoCAD 2006, it is carlson06.dwt. And for AutoCAD 2008, the Carlson template file is carlson07.dwt. After choosing the template, click the Open button at the lower-right. Next, you will either see the New Drawing Wizard dialog box, or you will be taken to a blank screen. Should you use the wizard, a new drawing name will need to be chosen in order to get to the next step.



If the wizard is in use, the following options will be available to you in the New Drawing Wizard dialog. The New command starts a new drawing using default settings defined in one of the Carlson .DWT template files, depending on the measurement system you've chosen. You cannot modify the surv.dwt or surviso.dwt templates. To start a new drawing based on a customized template, see Use a Template.



English: This option starts a new drawing based on the Imperial measurement system. The drawing is based on the surv.dwt template, and the default drawing boundary (the drawing limits) is 12 × 9 inches.
Metric: This option starts a new drawing based on the metric measurement system. The drawing is based on the surviso.dwt template, and the default drawing boundary (the drawing limits) is 429 × 297 millimeters.

The New command creates a new drawing, using the settings defined in a template drawing you select. Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views. Template drawings are distinguished from other drawing files by the .DWT file extension. They are normally kept in the template directory. Several template drawings are included with Carlson. You can make additional template drawings by changing the extensions of drawing file names to .DWT.

Remember that there are two methods that you can use to work on a Carlson drawing. One is the New command, and the other is the more generic Open command. If you need to open an existing drawing, use Open, also found in the File menu, and then choose an existing file name.

Pulldown Menu Location: File
Keyboard Command: new
Prerequisite: None

Open

This command allows you to open an existing drawing file. Carlson TakeOff displays the Select File dialog box (a standard file selection dialog box). Select a file and click Open.

Prerequisite: None

Keyboard Command: OPEN

Close

This command allows you to close the current drawing. Carlson TakeOff closes the current drawing if there have been no changes since the drawing was last saved. If you have modified the drawing, the program prompts you to save or discard the changes. You can close a file that has been opened in Read-only mode if you have made no changes or if you are willing to discard changes. To save changes to a read-only file, you must use the SAVEAS command.

Prerequisite: None

Keyboard Command: CLOSE

Save

If the drawing is named, Carlson TakeOff saves the drawing without requesting a file name. If the drawing is unnamed, the program displays the Save Drawing As dialog box (see SAVEAS) and saves the drawing with the file name you specify. If the drawing is read-only, use the SAVEAS command to save the changed file under a different name. This command allows you to save the drawing under the current file name or a specified name

Prerequisite: None

Keyboard Command: SAVE or QSAVE

Save As

This command allows you to save an unnamed drawing with a file name or renames the current drawing.

Carlson TakeOff displays the Save Drawing As standard file selection dialog box. Enter a file name and type. You can select any of the following file types:

- Carlson Software 2002/AutoCAD 2000 (∗.dwg)
- AutoCAD R14/LT 98/LT 97 Drawing (∗.dwg)
- AutoCAD R13/LT 95 Drawing (∗.dwg)
- Drawing Template File (∗.dwt)
- Carlson Software 2002 DXF (∗.dxf)
- AutoCAD R14/LT 98/LT 97 DXF (∗.dxf)
- AutoCAD R13/LT 95 DXF (∗.dxf)
- AutoCAD R12/LT2 DXF (∗.dxf)

Carlson TakeOff saves the file under the specified file name. If the drawing is already named, the program saves the drawing to the new file name. If you save the file as a drawing template, the program displays the Template Description dialog box, where you can provide a description for the template and set the units of measurement.

Saving a drawing in Release 14/LT 98/LT 97 format is subject to the following limitations:

- Hyperlinks are converted to Release 14 attached URLs.
- Database links and freestanding labels are converted to Release 14 links and displayable attributes.
- Database attached labels are converted to MText and leader objects, and their link information is not available. Attached labels are restored if you open the drawing in AutoCAD 2000 or later.
- Lineweight information is not available. Lineweights are restored if you open the drawing in AutoCAD 2000 or later.

Saving a drawing in Release 13/LT 95 format is subject to the following limitations:

- Lightweight polyline and hatch patterns are converted to R13 polylines and hatch patterns.
- Raster objects are displayed as bounding boxes. Raster objects are restored if the drawing is opened in AutoCAD 2000 or later.
- Draw order information is not applied for display or print.

- Xrefs that have been clipped with a boundary box are displayed in full as attached xrefs because Release 13 does not support xref clipping. Clipping is restored if the drawing is opened in AutoCAD 2000 or later.

Saving a drawing in Release 12/LT 2 DXF format is subject to the following limitations:

- Lightweight polylines and hatch patterns are converted to R12 polylines and hatch patterns.
- All solids, bodies, regions, ellipses, leaders, multilines, rays, tolerances, and xlines are converted to lines, arcs, and circles as appropriate.
- Groups, complex linetypes, OLE objects, and preview images are not displayed.
- Many objects are lost if you save a drawing as Release 12 and open it later in AutoCAD 2000 or later.

Prerequisite: None

Keyboard Command: SAVEAS

Plot

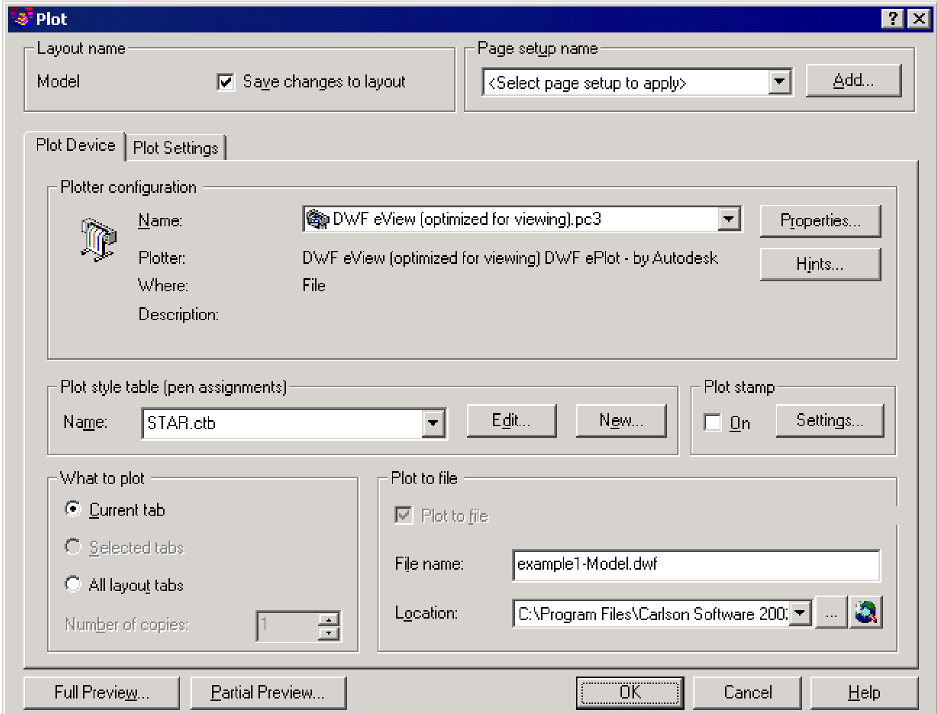
This command allows you to plot a drawing to a plotting device or file.

Carlson TakeOff displays the Plot dialog box. Choose OK to begin plotting with the current settings and display the Plot Progress dialog box.

1 The Plot dialog box includes the tabs, Plot Device and Plot Settings, and several options to customize the plot.

- Layout Name: This option displays the current layout name or displays "Selected layouts" if multiple tabs are selected. If the Model tab is current when you choose Plot, the Layout Name shows "Model."
- Save Changes to Layout: This option saves the changes you make in the Plot dialog box in the layout. This option is unavailable if multiple layouts are selected.
- Page Setup Name: This option displays a list of any named and saved page setups. You can choose to base the current page setup on a named page setup, or you can add a new named page setup by choosing Add.
- Add: This option displays the User Defined Page Setups dialog box. You can create, delete, or rename named page setups.

2 Under the Plot Device Tab you can specify the plotter to use, a plot style table, the layout or layouts to plot, and information about plotting to a file.



- Plotter Configuration: This field displays the currently configured plotting device, the port to which it's connected or its network location, and any additional user-defined comments about the plotter. A list of the available system printers and PC3 file names is displayed in the Name list. An icon is displayed in front of the plotting device name to identify it as a PC3 file name or a system printer.
- Properties: The option displays the Plotter Configuration Editor (PC3 Editor), where you can modify or view the current plotter configuration, ports, device, and media settings.
- Hints: This option displays information about the specific plotting device.
- Plot Style Table (Pen Assignments): This option sets the plot style table, edits the plot style table, or creates a new plot style table.
- Name: This option displays the plot style table assigned to the current Model tab or layout tab and a list of the currently available plot style tables. If more than one layout tab is selected and the selected layout tabs have different plot style tables assigned, the list displays "Varies."
- Edit: This option displays the Plot Style Table Editor, where you can edit the selected plot style table.
- New: This option displays the Add-a-Plot-Style-Table wizard, which you can use to create a new plot style table.
- Plot Stamp: This option places a plot stamp on a specified corner of each drawing and/or logs it to a file.
- On: This options turns on plot stamping.
- Settings: This option displays the Plot Stamp dialog box, where you can specify the information you want applied to the plot stamp, such as drawing name, date and time, and plot scale.
- What to Plot: This field defines the tabs to be plotted.
- Current Tab: This option plots the current Model or layout tab. If multiple tabs are selected, the tab that shows its viewing area is plotted.
- Selected Tabs: This option plots multiple preselected Model or layout tabs. To select multiple tabs, hold down CTRL while selecting the tabs. If only one tab is selected, this option is unavailable.

• All Layout Tabs: This option plots all layout tabs, regardless of which tab is selected.

• Number of Copies: This option denotes the number of copies that are plotted. If multiple layouts and copies are selected, any layouts that are set to plot to a file or AutoSpool produce a single plot.

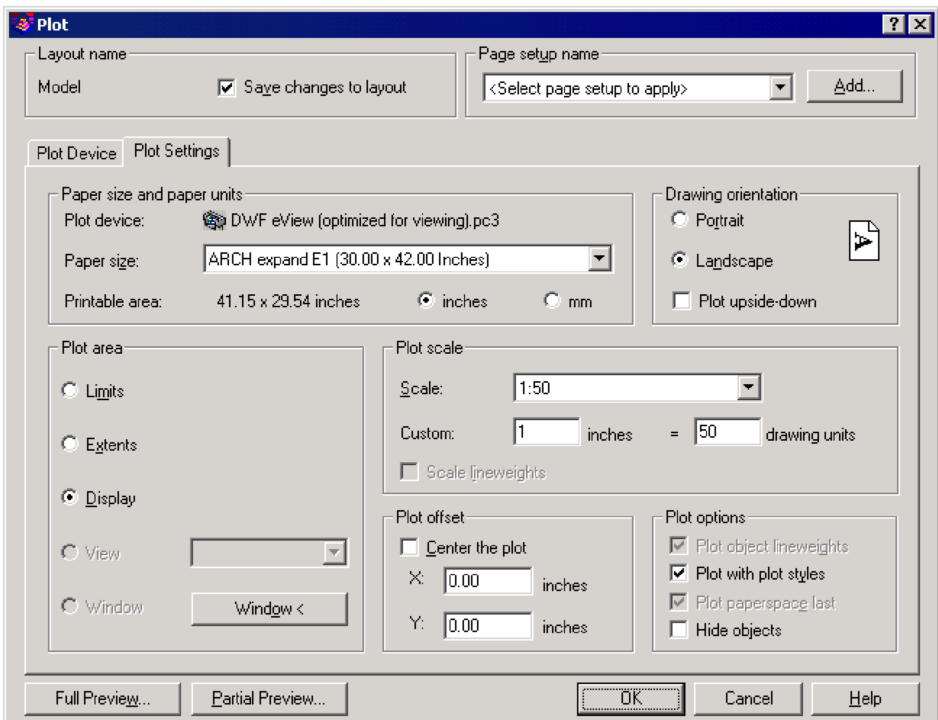
• Plot to File: This option plots output to a file rather than to the plotter.

• File Name: This option specifies the plot file name. The default plot file name is the drawing name and the tab name, separated by a hyphen, with a .plt file extension.

• Location: This option displays the directory location where the plot file is stored. The default location is the directory where the drawing file resides.

• [...]: This option displays a standard Browse for Folder dialog box, where you can choose the directory location to store a plot file.

3 Under the Plot Settings Tab you specify paper size, orientation, plot area and scale, offset, and other options.



• Paper Size and Paper Units: This field displays standard paper sizes available for the selected plotting device. Actual paper sizes are indicated by the width (X axis direction) and height (Y axis direction). If no plotter is selected, the full standard paper size list is displayed and available for selection. A default paper size is set for the plotting device when you create a PC3 file with the Add-a-Plotter wizard. The paper size you select is saved with a layout and overrides the PC3 file settings. If you are plotting a raster image, such as a BMP or TIFF file, the size of the plot is specified in pixels, not in inches or millimeters.

• Plot Device: This field displays the name of the currently selected plot device.

• Paper Size: This field displays a list of the available paper sizes.

• Printable Area: This field displays the actual area on the paper that is used for the plot based on the current paper size.

• Inches: This option allows you to specify inches for the plotting units.

• MM: This option allows you to specify millimeters for the plotting units.

- Drawing Orientation: This option specifies the orientation of the drawing on the paper for plotters that support landscape or portrait orientation. You can change the drawing orientation to achieve a 0-, 90-, 180-, or 270-degree plot rotation by selecting Portrait, Landscape, or Plot Upside-Down. The paper icon represents the media orientation of the selected paper. The letter icon represents the orientation of the drawing on the page.
- Portrait: This option orients and plots the drawing so that the short edge of the paper represents the top of the page.
- Landscape: This option orients and plots the drawing so that the long edge of the paper represents the top of the page.
- Plot Upside-Down: This option orients and plots the drawing upside down.
- Plot Area: This option specifies the portion of the drawing to be plotted.
- Layout: This option plots everything within the margins of the specified paper size, with the origin calculated from 0,0 in the layout. Available only when a layout is selected. If you choose to turn off the paper image and layout background on the Display tab of the Options dialog box, the Layouts selection becomes Limits.
- Limits: This option plots the entire drawing area defined by the drawing limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option. Available only when the Model tab is selected.
- Extents: This option plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted. TakeOff may regenerate the drawing to recalculate the extents before plotting.
- Display: This option plots the view in the current viewport in the selected Model tab or the current paper space view in the layout.
- View: This option plots a previously saved view. You can select a named view from the list provided. If there are no saved views in the drawing, this option is unavailable.
- Window: This option plots any portion of the drawing you specify. If you select Window, the Window button becomes available. Choose the Window button to use the pointing device to specify the two corners of the area to be plotted or enter coordinate values.
- Plot Scale: This option controls the plot area. The default scale setting is 1:1 when plotting a layout. The default setting is Scaled to Fit when plotting a Model tab. When you select a standard scale, the scale is displayed in Custom.
- Scale: This option defines the exact scale for the plot. The four most recently used standard scales are displayed at the top of the list.
- Custom: This option creates a custom scale. You can create a custom scale by entering the number of inches or millimeters equal to the number of drawing units.
- Scale Lineweights: This option scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of printed objects and are plotted with the linewidth size regardless of the plot scale.
- Plot Offset: This field specifies an offset of the plotting area from the lower-left corner of the paper. In a layout, the lower-left corner of a specified plot area is positioned at the lower-left margin of the paper. You can offset the origin by entering a positive or negative value. The plotter unit values are in inches or millimeters on the paper.
- Center the Plot: This option automatically calculates the X and Y offset values to center the plot on the paper.
- X: This field specifies the plot origin in the X direction.
- Y: This field specifies the plot origin in the Y direction.
- Plot Options: This field specifies options for lineweights, plot styles, and the current plot style table. You can select whether lineweights are plotted. By selecting Plot with Plot Styles, you plot using the object plot styles that are assigned to the geometry, as defined by the plot style table.

- Plot object lineweights: This option plots lineweights.
- Plot with Plot Styles: This option plots using the plot styles applied to objects and defined in the plot style table. All style definitions with different property characteristics are stored in the plot style tables and can be easily attached to the geometry. This setting can replace pen mapping in earlier versions of AutoCAD.
- Plot Paperspace Last: This option plots model space geometry first. Paper space geometry is usually plotted before model space geometry.
- Hide Objects: This option plots layouts with hidden lines removed for objects in the layout environment (paper space). Hidden line removal for model space objects in viewports is controlled by the Viewports Hide property in the Object Property Manager. This is displayed in the plot preview, but not in the layout.
- Full Preview: This option displays the drawing as it will appear when plotted on paper. To exit the print preview, right-click and choose Exit.
- Partial Preview: This option quickly shows an accurate representation of the effective plot area relative to the paper size and printable area. Partial preview also gives advance notice of any warnings that you might encounter when plotting. The final location of the plot depends on the plotter. Changes that modify the effective plot area include those made to the plot origin, which you define under Plot Offset on the Plot Settings tab. If you offset the origin so much that the effective area extends outside the preview area, the program displays a warning.

Prerequisite: None

Keyboard Command: PLOT

Recover

Function

This command opens a drawing file and scans it for errors. Use this command if Carlson TakeOff crashes while using the regular **Open** command.

Prerequisite: none

Keyboard Command: recover

Audit

Function

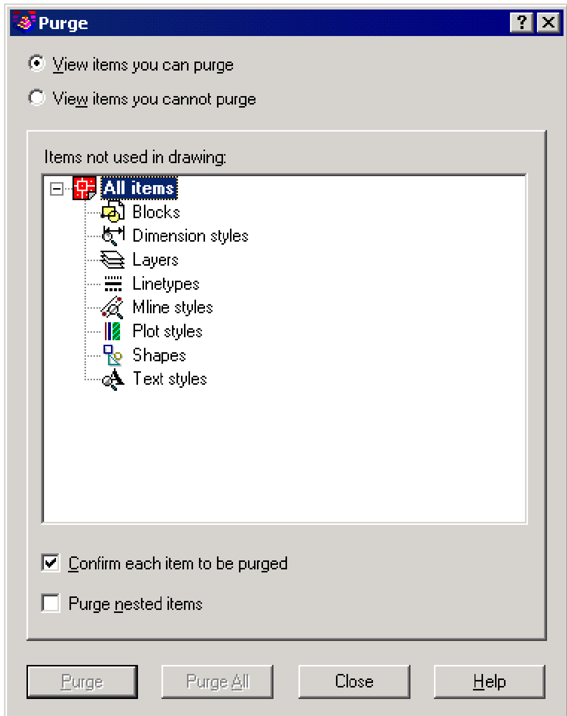
This command scans your current drawing and looks for any corruption and has the option to fix any errors.

Prerequisite: none

Keyboard Command: audit

Purge

Displays a tree view summary of all named objects that can and can't be purged in the current drawing. The View Items You Can Purge and View Items You Cannot Purge options toggle the dialog box display, showing different options and tree view summaries.



Items Not Used in Drawing: Displays a tree view of all named object categories (blocks, layers, and so on) in the current drawing. A plus sign appears next to the object category names that you can purge. Clicking the plus sign or double-clicking an object category expands the tree view, displaying all unused named objects that exist for the category. To purge all unused named objects, select All Items in the tree view, and choose Purge All. To purge a specific named object category, select the category in the tree view, and choose Purge.

Confirm Each Item to Be Purged: Displays the Verify Purge dialog box when you purge an item.

Purge Nested Items: Removes all unused named objects from the drawing even if they are contained within or referenced by other unused named objects. The Verify Purge dialog box is displayed, and you can cancel or confirm the items to be purged.

Prerequisite: None

Keyboard Command: purge

Store Project Archive

Function

This command creates an archive of the current project. The archive contains the drawing file (.dwg) and all the associated data file such as the surfaces and layer target definitions. This archive can be used as a backup for the project or as a way to transfer the project to another computer. The format of the archive file is a standard .zip file which can be used by WinZip.

When this command is run, the program will ask for a file name of the archive to create. Enter a name and pick the Save button. The number of files stored to the archive is reported at the command line.

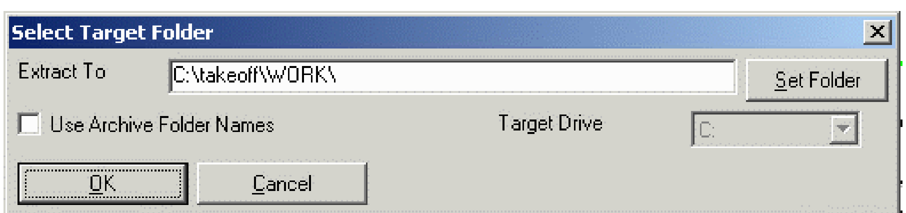
Prerequisite: an active Takeoff project

Keyboard Command: zip_project

Extract Project Archive

Function

This command reads the project files from an archive created by the Store Project Archive command. The archive contains the drawing file (.dwg) and all the associated data file such as the surfaces and layer target definitions. Since the archive contains the drawing file, you should not have the same project drawing open in Takeoff while extracting the archive. The format of the archive file is a standard .zip file which can be used by WinZip.



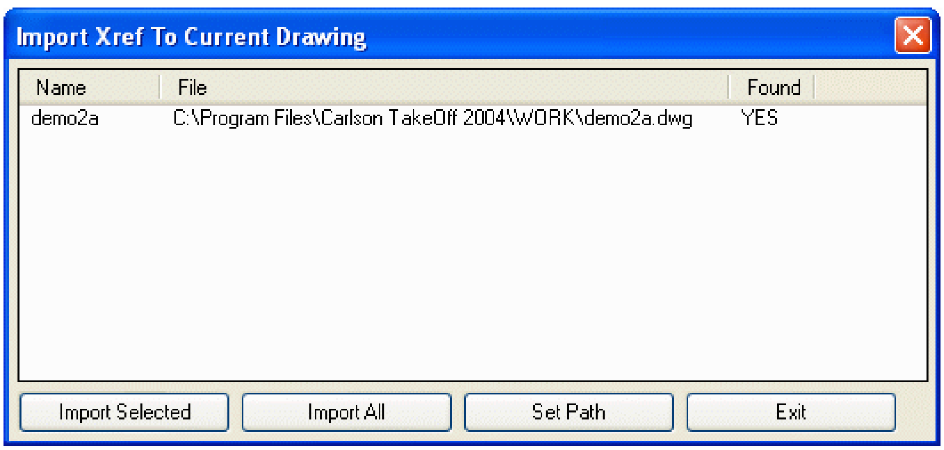
The command starts by prompting for the archive file to open. Then there is a dialog with extraction options. You can either extract the files to the specified folder or use the folder names stored in the archive. When using the archive folders, there is a setting to choose the target drive because the archive has the folder names but not the drive names.

Prerequisite: a Takeoff archive file

Keyboard Command: unzip_project

Import Xref to Current Drawing

This command allows you to import external reference files (Xrefs) into the current drawing. Before Xrefs are imported, the drawing data from the Xrefs can be viewed but not modified. This import routine has a simplier method for importing than the Xref Manager command. A list is shown of the Xrefs that are attached to the current drawing. If the Xref file is not found, you can pick the Set Path button to locate the drawing file. To import an Xref, highlight the file name and Pick Import.

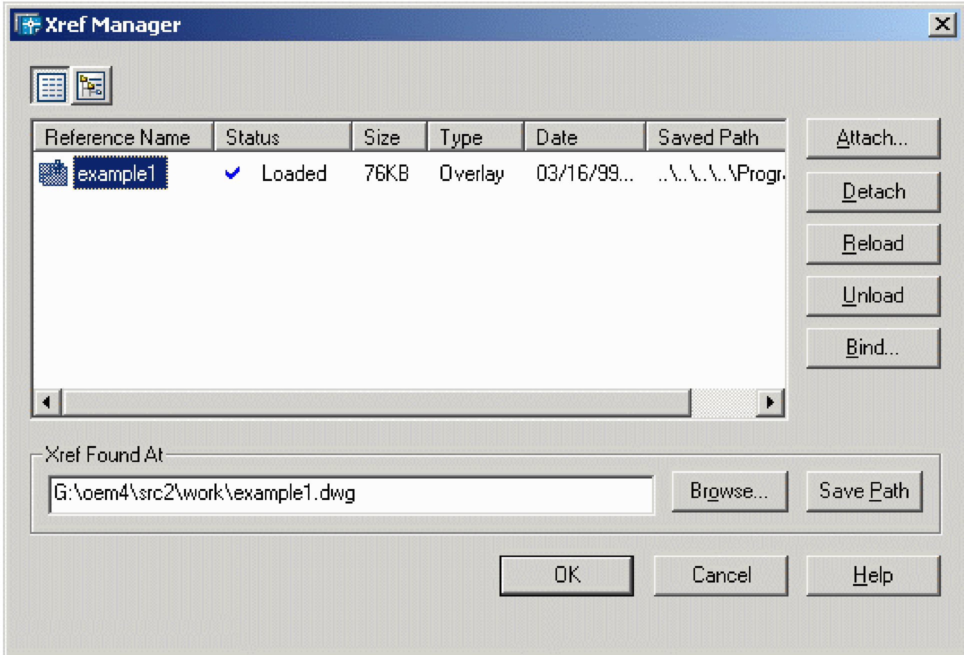


Prerequisite: files to import

Keyboard Command: import_xref

Xref Manager

Attaches, overlays, lists, binds, detaches, reloads, unloads, renames, and modifies paths to external references (Xrefs) in the current (or host) drawing. Displays the Xrefs in the drawing in a tree view or a list view. You can use the F3 and F4 keys to switch between list view and tree view.



List View: Displays a flat listing of the attached Xrefs and their associated data. You can sort the list of references by name, status, type, file date, file size, or the saved path and file name.

Reference Name: Lists the names of the Xrefs as stored in the definition table for the drawing. Status: Shows whether the Xref is loaded, unloaded, unreferenced, not found, unresolved, orphaned, or marked for unloading or reloading.

- Loaded: Currently attached to the drawing.
- Unloaded: Marked to be unloaded from the drawing once the Xref Manager is closed.
- Unreferenced: Attached to the drawing but erased.
- Not Found: No longer exists in the valid search paths.
- Unresolved: Cannot be read by AutoCAD.
- Orphaned: Attached to another Xref that is unreferenced, unresolved, or not found.

Size: Shows the file size of the corresponding reference drawing. The size is not displayed if the Xref is unloaded, not found, or unresolved.
Type: Indicates whether the Xref is an attachment or an overlay.
Date: Displays the last date the associated drawing was modified. This date is not displayed if the Xref is unloaded, not found, or unresolved.
Saved Path: Shows the saved path of the associated Xref (this is not necessarily where the Xref is found).
Tree View: Displays a hierarchical representation of the Xrefs, displaying the relationships between Xref definitions. Tree view shows the level of nesting relationship of the attached Xrefs, whether they are attached or overlaid, and whether they are loaded, unloaded, marked for reload or unload, or not found, unresolved, or unreferenced.
Attach: Displays the External Reference dialog box if an external reference is selected or displays the Select Reference File dialog box if no external reference is selected.
Detach: Detaches one or more Xrefs from your drawing, erasing all instances of a specified Xref and marking the Xref definition for deletion from the symbol table. Only the Xrefs attached or overlaid directly to the current drawing can be detached; nested Xrefs cannot be detached. Carlson TakeOff cannot detach an Xrefreferenced by

another Xref or block.
Reload: Marks one or more Xrefs for reloading. This option rereads and displays the most recently saved version of the drawing.

Unload: Unloads one or more Xrefs. Unloaded Xrefs can be easily reloaded. Unlike detaching, unloading does not remove the Xref permanently. It merely suppresses the display and regeneration of the Xrefdefinition to improve performance.

Bind: Displays the Bind Xrefs dialog box Xref. The Bind option makes the selected Xrefand its dependent symbols (such as blocks, teXreft styles, dimension styles, layers, and linetypes) a part of the current drawing.
Found At: Displays the full path of the currently selected Xref. This is where the Xref is actually found and is not necessarily the same as the saved path.
Browse: Displays the Select New Path dialog boXref (a standard file selection dialog boXref), in which you can select a different path or file name.

Save Path: Saves the path, as it appears in XrefFound At, to the currently selected Xref.

Prompts

Command: _Xref
Overlay Xref "example1": ..\..\..\..\Program Files\Carlson TakeOff 2004\WORK\example1.dwg
"example1" loaded: G:\oem4\src2\work\example1.dwg

Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]:
Command: Specify opposite corner:
Select objects: Enter

Prerequisite: multiple files

Keyboard Command: Xref

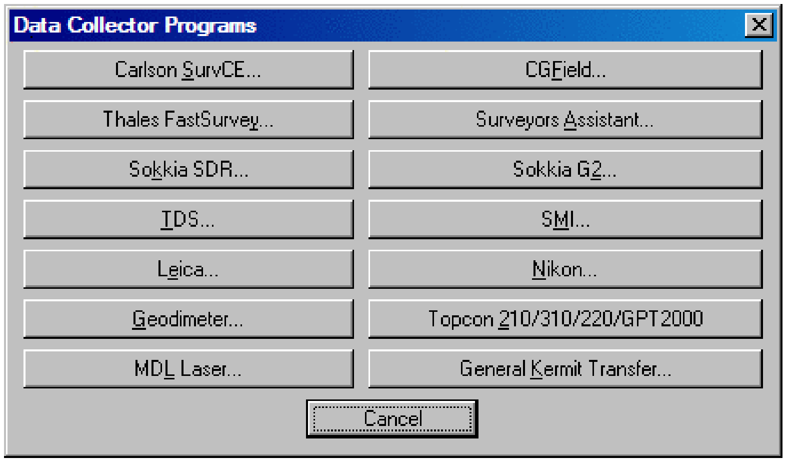
Data Collectors

Function

This command does two main functions for a variety of popular data collectors. First, this command transfers (uploads and downloads) data between the data collector and Carlson. Second, this command converts data formats between the data collector format and the Carlson format. So, if you already have the data file on the computer, you can skip the transfer function and just perform the conversion function.

The transfer function does the conversion at the same time. In most cases, the download from the data collector produces a raw (.RW5) file (field notes) and/or a coordinate (.CRD) file (coordinate points). Several of the download

programs have an option to automatically run the *Edit-Process Raw Data File* command after downloading raw data. You can also send, or upload, a coordinate (.CRD) file. The dialog shown here appears when the menu command is selected.



Carlson SurvCE: For Carlson Software data collection programs SurvCE and SurvStar. This button produces the SurvCOM dialog and program.
CG Field: For CG Field programs.
Thales FastSurvey: For Thales and FastSurvey intstruments.
Surveyors Assistant: For data collectors running Surveyors Assistant software (Corvallis MC2, MC5 and Pentax SC5).
Sokkia SDR: For SDR2 through SDR 33 and other collectors that have a SDR format like the Trimble.
Sokkia G2: Specifically for the SDR2.
TDS: For data collectors that use TDS software (Ranger, HP48, HP95, Husky FS-2 & FS-3, Corvallis MC-V and TOPCON FS2, FC95 and FC48).
SMI: For SMI data collectors on the HP48.
Leica: For Leica GIF-10 module and Leica instruments.
Nikon: For Nikon DTM and DR-48 total stations.
Geodimeter: For the Geodimeter Geodat collector.
Topcon 210/310/220/GPT2000: Supports these Topcon models.
MDL Laser: For MDL Laser instruments.
General Kermit Transfer: For general transferring using Kermit.

Carlson SurvCE

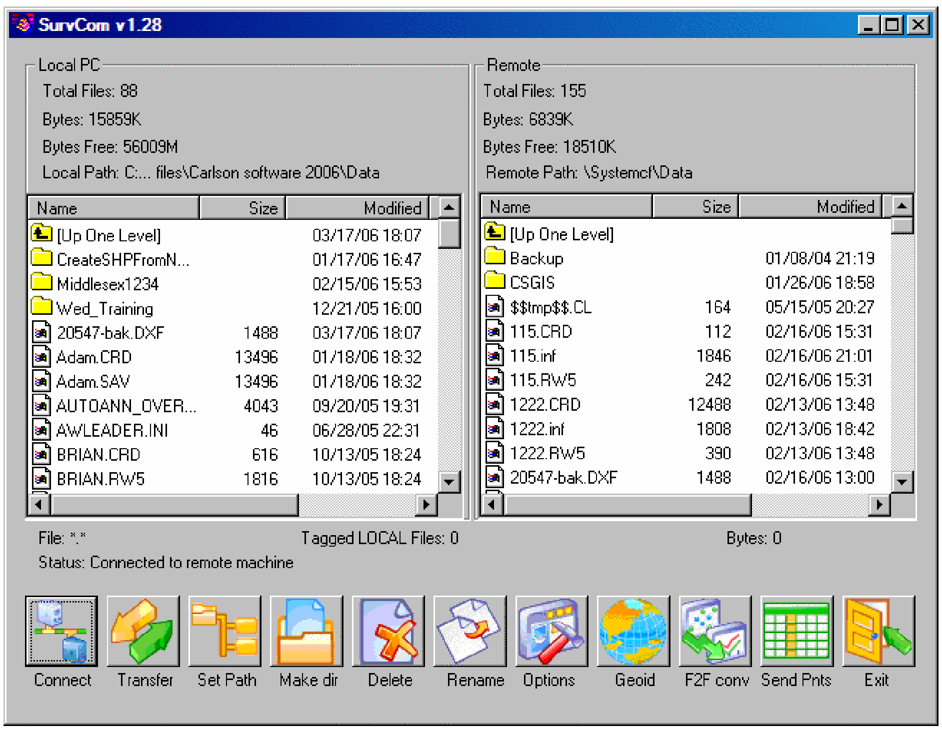
Note: In the following text, the term SurvCE will apply to SurvCE, SurvStar, and Sokkia G2

Connect the serial cable. Select Data Transfer from the on the handheld. Choose Carlson/Carlson Survey Download. This leads to a File Transfer screen on SurvCE, which says "Awaiting Connection". All the action is on the PC side. There is no time delay in this handshake. It will wait for the PC program to catch up. When you connect the cable from SurvCE to the PC, Microsoft ActiveSync may interfere and say "Connect to PC?" If you get this question, say No or otherwise terminate the Microsoft ActiveSync linkage. Start the Carlson portion of this link by choosing Survey, Data Collectors, then the SurvCE option. If connection is automatically established, SurvCE will display, "Connected to PC".

If only the left side of the screen displays data, then you do not yet have a connection. Press the Connect button located at the bottom left of the file transfer dialog. The transfer program will respond with Retrieving File List. Once the file list has been retrieved, the left side of the dialog box will show files located in the specified path on the PC and the right side of the dialog displays the files located in the designated path on the remote. You can change directories by scrolling to the top of the file list and choosing Up One Level (just like in Windows).

To transfer one or more files, simply select or highlight the desired files and select the transfer button. More than one file can be transferred from the remote to the PC or from the PC to the remote during the transfer process. Standard Windows selection options apply. For example, selecting one file and then while pressing the shift key on the PC, selecting another file deeper on the list will select all the files in between the first and last selected. You can also select the first file to transfer and press and hold down the shift key and use the down arrow to specify the range of files to transfer. Pressing and holding the control key on the keyboard allows for the selection of multiple files in any selection order, by picking the files with the left mouse button.

After the files have been selected, press the transfer button. When the transfer is complete, the program will return a "Transfer Complete" message, and will then proceed to update the file lists on the PC and the Remote.



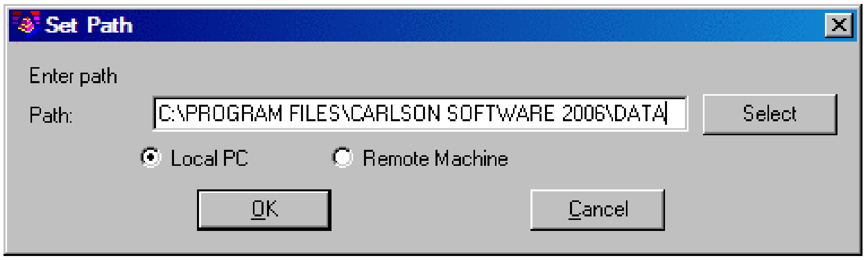
The following information describes the buttons on the bottom row of the SurvCOM dialog box. The button name is on the left in bold:

Connect: After selecting Data Transfer in SurvCE, press this button to start the connection. Once connection is made, the status line on the
file transfer utility dialog box will show Connected to the remote machine.

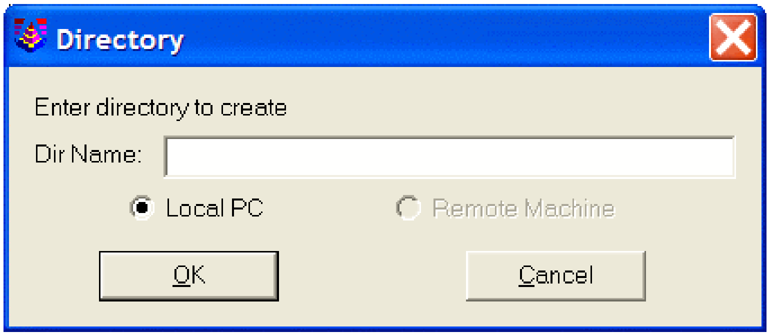
Transfer: Pressing this button transfers selected files from either the Remote to the PC, or the PC to the Remote.

Set Path: This option allows for the specification of the desired source and destination drives and folders for both the PC and the Remote device. For example, if you were downloading, or copying files from the Remote device to the PC, to specify a source path on the remote device, select the Remote Machine toggle and then type in the desired path in the path field. To specify a destination path on the PC, select the Local PC toggle and type in the

desired path the path field. When a change to either path is made, the transfer utility will retrieve a new file list from the specified paths.

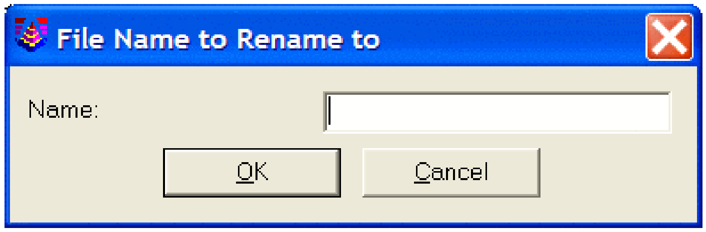


Make dir: This option allows for creation of directories on both the PC and the Remote device. Specify the hardware on which to create the directory and then enter the directory name.

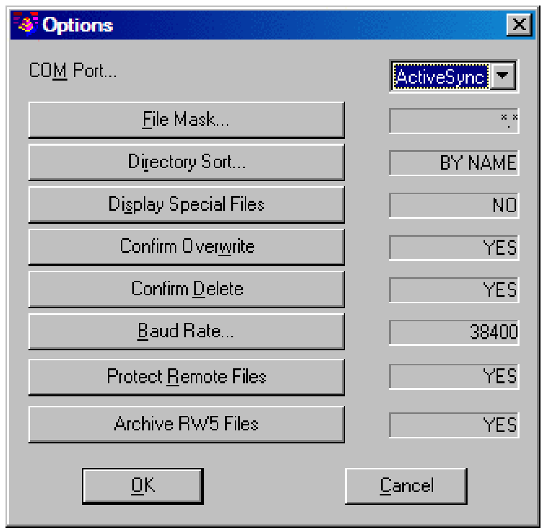


Delete: This option allows you to delete the tagged files. To delete a file, select the file to delete by clicking on the file, press the delete button at the bottom of the dialog. Confirm deletion by selecting the appropriate response on the Delete File dialog.

Rename: To rename a file, click on the file to rename and select the rename button at the button of the dialog. On the dialog that displays type in the new name and press the OK button.

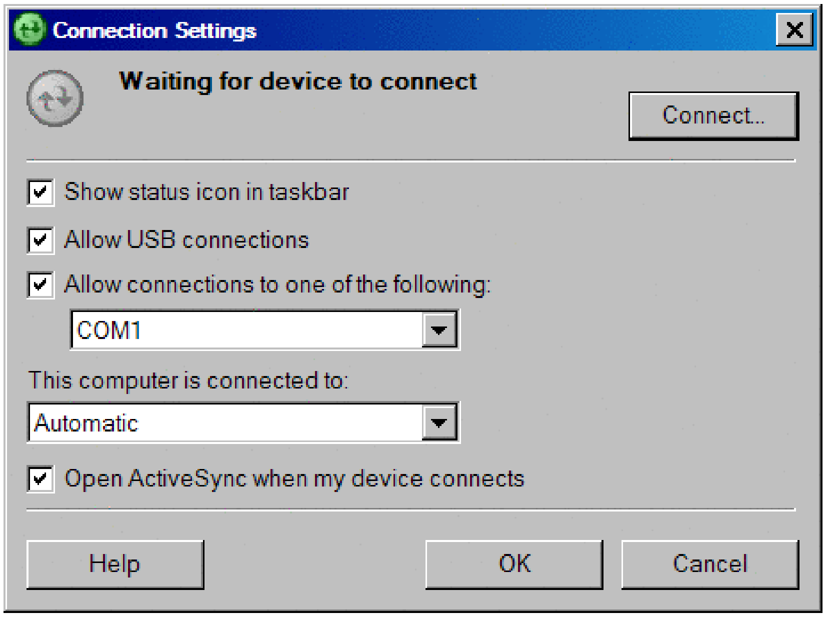


Options: This command allows you to set various options for data transfer. The dialog shown below will appear.

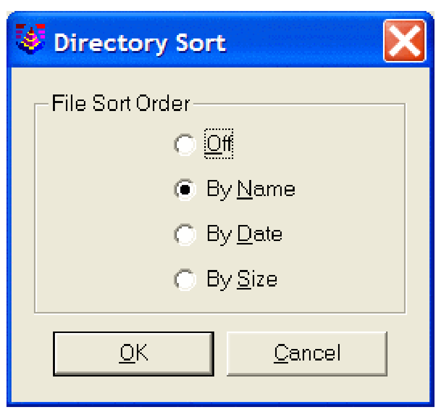


Com Port: You must select which com port on the PC to use.

If you are transferring data via a USB port, set the com port to ActiveSync, see the Options section below for procedures to change com ports. To transfer data using an USB port a connection between the Remote and PC using ActiveSync is required. In ActiveSync verify that the "Connect Settings" have been set to "Allow serial cable or infrared connection to this Com port" and Allow USB connection with this desktop computer. This will allow for connection using an USB port or a COM port connection. Both will use ActiveSync to transfer data between devices.



File Mask: You must select a file filtering syntax. This filter allows for the setting of specific file types to display. For example if you only wanted to see CRD files the filter would be ∗.CRD.
Directory Sort: You must select how to sort the list of files.



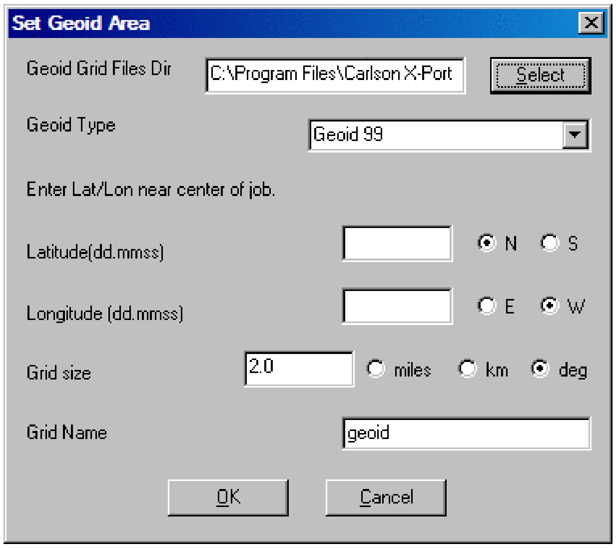
Display Special Files: Toggle whether or not you should see special files.
Confirm Overwrite: Check this to confirm before overwriting files.
Baud Rate: You must choose the baud rate for transferring data.
Protect Remote Files: Check this to protect files on the mobile device.
Archive RW5 Files: With this option set to YES, when downloading rw5 files, a second copy of the file will be made with a .SC5 extension to serve as an archive of the original rw5 file.

Geoid: This command will carve out a portion of the Geoid 99, EGM96, Canadian CGC2000, Canadian HT2.0, Canadian HT 1.01, Australian GDA94, Great Britain OSG-MO2 and Geoid 2003 grid files, and send it to SurvCE. Since these geoid grids are very large, this carves out a precise portion of it and avoids overloading the memory on the remote device running SurvCE. You will be prompted for the directory on the PC of the source Geoid grid file,

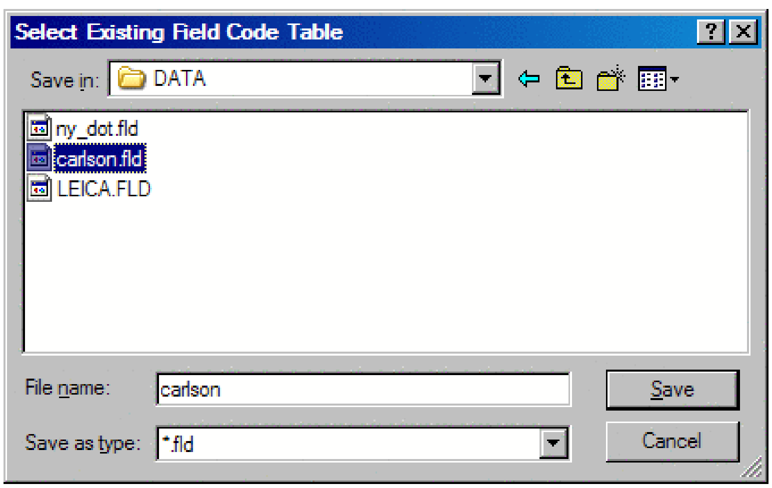
the approximate latitude and longitude of the job, and the size of the area desired in miles, kilometers or degrees of latitude and longitude. To define a Geoid area, make sure that this criteria is met:

1. Specify the location of the geoid grid files.
2. Specify the geoid type.
3. Enter the latitude and longitude near the center of the job area.
4. Specify the Grid size either in miles, km (kilometers), or deg (degrees).
5. Name the grid file.

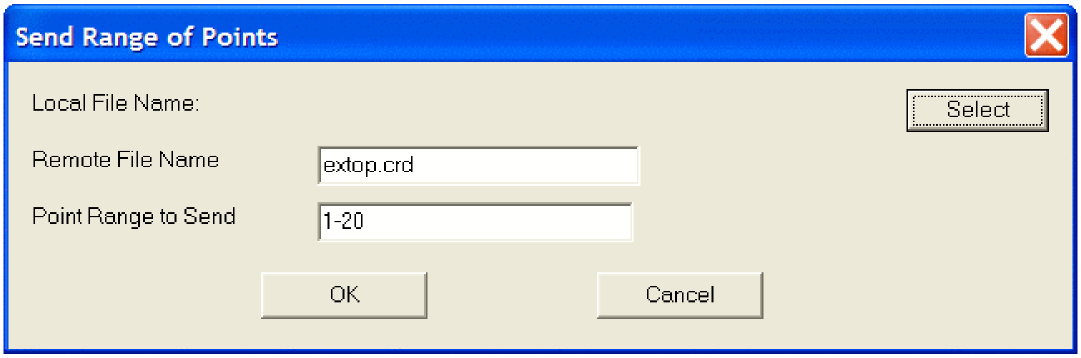
The file will be transferred to the data collector and place in the appropriate place for use.



F2F conv: This converts the more thorough and detailed Carlson field code file (for field-to-finish work, ∗.FLD) to the more simplified Feature Code List that runs in SurvCE (∗.FCL). The Feature Code List in SurvCE (not SurvStar or Field) handles Linework (on or off), Line Type (2D or 3D), Layer (= Code) and Full Text (Description). Select the Carlson field code (∗.FLD) to convert, the conversion takes place and the file is transferred and located in the correct location for use in the data collector.



Send Pnts: This option allows for the uploading of a user specified point number range out of the selected crd file to unload. Use the Select button to specify the crd file to upload. The Remote File Name will default to the name of the crd file selected to upload. You can change this name if needed. Specify the Point Range to Send and select the OK button.



Exit: This command will exit the File Transfer Utility

The following information describes the buttons on the Data Collection Programs dialog box that come after the Carlson SurvCE buttom, moving from left to right and then from top to bottom. The command/button name is on the far left margin, in bold:

CG Field

To transfer data to and from data collectors using CGField software, first make sure that the Baud Rate is set to 9600 and the Parity is set to NONE then follow the steps outlined below.

Receiving a Coordinate File from CGField

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 4, "Send Coords"
3) Select the Coordinate file to send.

Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

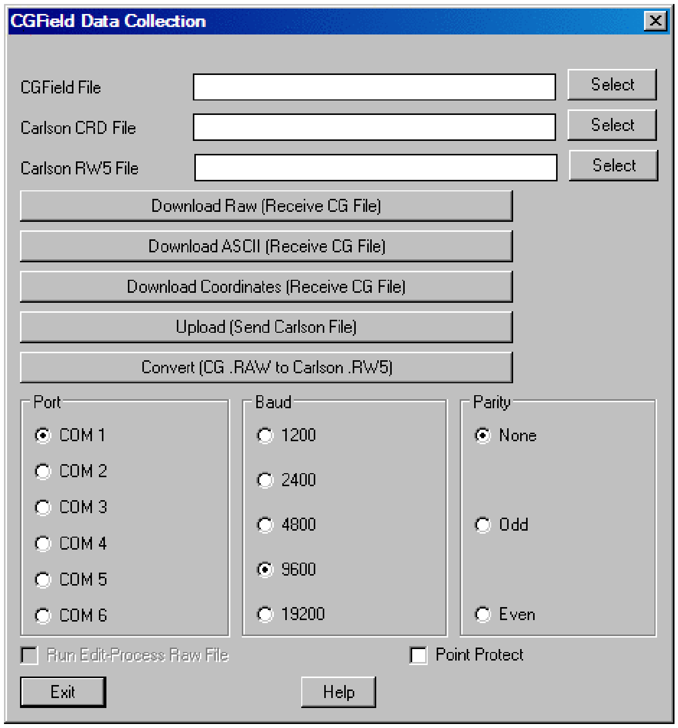
Press the "Download Coordinates" button to ready Carlson to receive the file.
Stop here in Carlson and go back to CGField to complete the transfer process.

CGField:

Select the points to send

1) For All points
2) To select Blocks of points.
3) From .PTS file (the set of points in a Batch Point File).

The coordinates will be transferred. After the transfer is complete, you will be asked for the CRD file name. The C&G CRD file will automatically be converted to a Carlson CRD file. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.



Receiving a Raw Data File from CGField

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 2, "Send Raw Data". Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

Press the "Download Raw" button to ready Carlson to receive the file. Stop here in Carlson and go back to CGField.

CGField:

Select the raw data file to be sent. The transfer will begin.

The C&G .RAW file will be transferred and saved in the data folder. After the transfer is complete, you will be asked for the RW5 file name. The RAW file will be automatically converted to a Carlson RW5 file.

Receiving an ASCII file from CGField

This will allow you to transfer a C&G report file (RPT) or an ASCII NEZ file to Carlson.

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 6, "Send ASCII". Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

Press the "Download ASCII" button to ready Carlson to receive the file. Stop here in Carlson and go back to CGField.

CGField:

Select the ASCII file to send.

After the transfer is complete, you will see the file in the Carlson editor. You can then select FILE and SAVE (or SAVEAS) to save the ASCII file.

Sending a Coordinate File to CGField

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 3, "Receive Coords" to ready the data collector. Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

1) Press the "Upload (Send Carlson File)" button.
2) Select the Coordinate file.
3) Select the points to send.
4) Press the "Start Transfer" button.

CGField:

Carlson will send the file name to CGField and a coordinate file with the same name will be automatically created or opened in CGField.

If the file exists you will be asked how you want to handle duplicate points:

1) Overwrite
2) Don't Overwrite
3) Ask for each Point

The point transfer will begin.

Convert CG .RAW to Carlson .RW5

This utility allows you to convert a C&G raw data file to a Carlson raw data file. Select the C&G .RAW file to convert. Then enter the file name of the destination Carlson RW5 file.

Thales/FastSurvey You will be taken directly to the SurvCOM dialog, similar to the Carlson SurvCE process.

Surveyor's Assistant

Download
From the Surveyor's Assistant data collector, go to the Transfer routine from the main menu. Fill out the transfer screen as follows:

Direction: OUTPUT
Format: LIETZ
Data: Coordinate or All Data
Port: COM1 or COM2 Ckh Hold: NO
Protocol: NONE

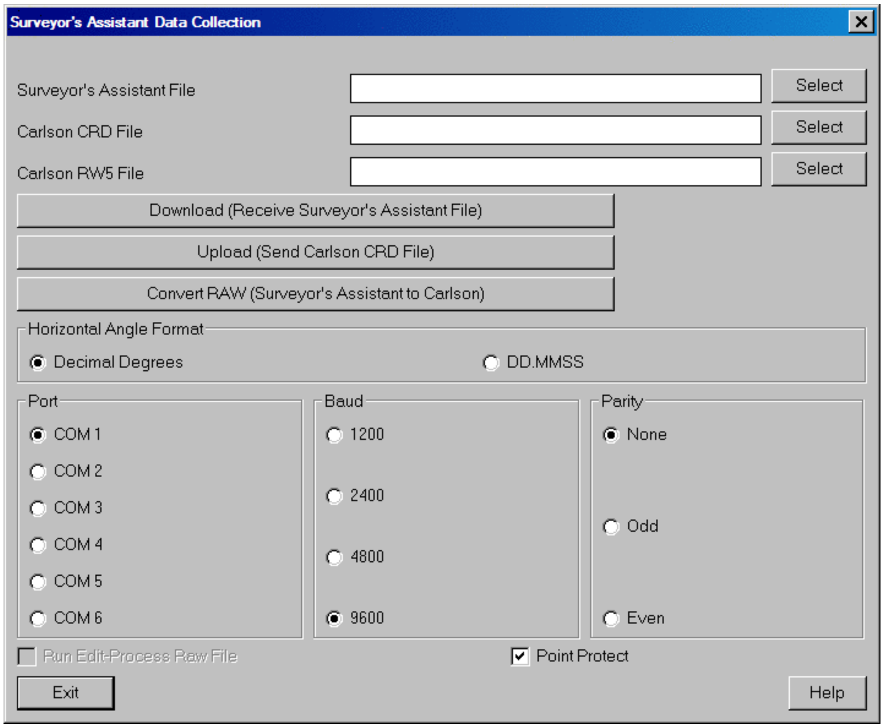
You should also check the settings under the PORT menu. Typical port settings are baud=9600, parity=none, data=8, stop=1 and handshake=XON/XOFF. Now in Carlson, run *Data Collection* in the Survey menu and choose Surveyor's Assistant. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to Surveyor's Assistant and press GO. The file transfer should now go. If the All Data option is used, then the Leitz format will contain both coordinate and raw data. The coordinate data is converted to a Carlson coordinate (.CRD) file and the raw data is converted to a Carlson raw data (.RW5) file. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

Point data from the Carlson coordinate (.CRD) file can be uploaded into the Surveyor's Assistant. First go to the Transfer routine on the main menu. Fill out the screen as follows:

Direction: INPUT
Format: LEITZ
Port: COM1 or COM2
Protocol: NONE

Go back to Carlson and choose Surveyor's Assistant from the *Data Collection* command in the Survey menu. Check that the COM port and baud rate are set correctly. In the Carlson dialog, pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Before clicking the OK button for range of points, go to the Surveyor's Assistant and hit the GO function key. The Surveyor's Assistant is now waiting to receive so return to Carlson and click OK on the range of point dialog. The file transfer should now go.



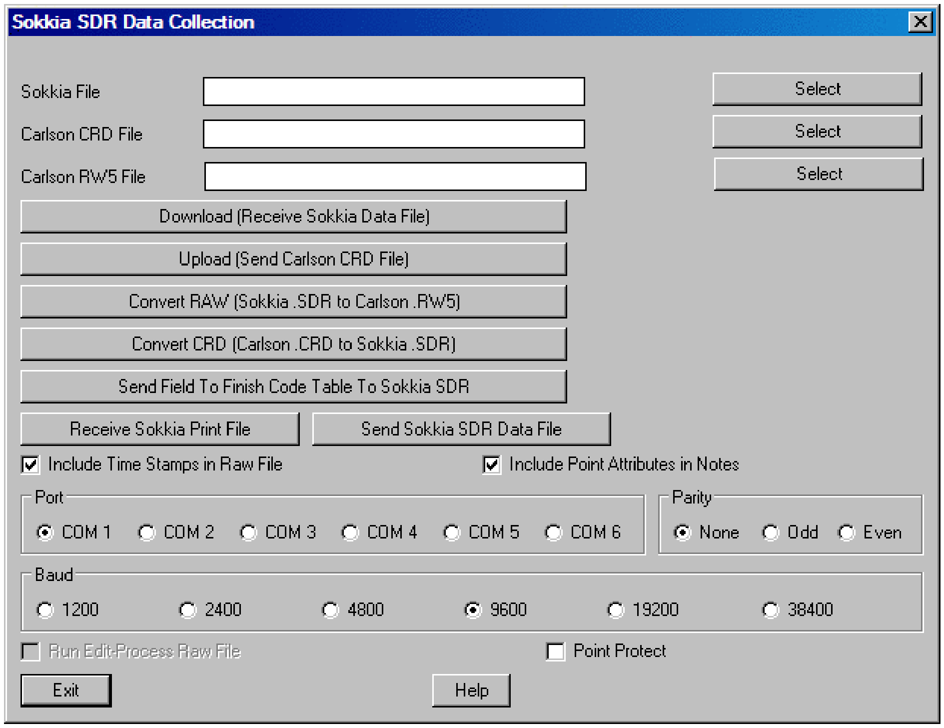
Sokkia SDR

This routine applies to the Sokkia SDR-20, SDR-22, SDR-31 and SDR-33 as well as other collectors that have SDR format transfer such as the Trimble and C & G.

Download

From the SDR data collector, go to the Communications routine from the main menu. Choose Data Format SDR. Next hit the Send function key. Then choose Select Jobs. From the list of jobs, highlight the job to transfer and set it to Yes with the arrow keys. Now in Carlson, run *Data Collection* in the Survey menu and choose Sokkia/SDR. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to SDR and press OK. The file transfer should now go. The SDR format contains both coordinate and raw data. The coordinate data is converted to a Carlson coordinate (.CRD) file and the raw data is converted to a Carlson raw data (.RW5) file. The original SDR transfer file is stored on the computer as a RAW file. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

The SDR-33 has different modes for storing and transferring data. In coordinate mode, the download will create points in the coordinate (.CRD) file and the raw data (.RW5) file will only contain some basic header lines. In the raw data mode, the download will create all the measurement data in the raw file and no points will be created in the coordinate (.CRD) file. For this raw data mode, you will need to run *Edit-Process Raw Data File* in the Survey menu to calculate the points from the raw data. The third mode in the SDR-33 creates both raw data in the raw data (.RW5) file and points in the coordinate (.CRD) file. The Include Time Stamps in Notes option sets whether all the date-time records for each point are put in the raw data (.RW5) file as description records. The Include Point Attributes in Notes option will store SDR code 13(AT) codes to the point note (.NOT) for the coordinate (.CRD) file.



Upload
Point data from the Carlson coordinate (.CRD) file can be uploaded into the SDR. First go to the Communications routine on the SDR main menu. Choose Data Format SDR. Go back to Carlson and choose Sokkia/SDR from the *Data Collection* command in the Survey menu. Check that the COM port and baud rate are set correctly. In the Carlson dialog, pick the Select File button next to the Carlson CRD File edit box and choose the coordinate (.CRD) file to send. Then click the Upload button. Then a Sokkia Options dialog appears for setting the job parameters for the file to be created on the collector. Be sure to choose the Distance Unit that matches your coordinate (.CRD) file (meters, US feet or international feet). Click OK and the next dialog now allows you to specify the range of point numbers to upload. Before clicking the Start Transfer button for range of points, go to the SDR and hit the Receive function key. The SDR is now waiting to receive so return to Carlson and click Start Transfer on the range of point dialog. The file transfer should now go.

Communication Settings
Besides matching the baud rate between Carlson and the collector, make sure that the collector is set to word length of 8 and 1 stop bit under the communication settings.

Print File
The Receive Sokkia Print File downloads a print report from the SDR33 data collector. This file is only used for printing report purposes in Carlson. This file is not used by Carlson to generate coordinate (.CRD) files or raw files. The first step is to choose Data format=Printed in the Communications menu of the SDR33. Next pick the Receive Print File button in Carlson. Then on the SDR33 choose the Send function and select a job to send. At this point the file is transferred. After downloading, the job report is displayed in the Carlson standard report viewer.

Example of Sokkia Printed Format:

```
SDR33 V04-04.25 (C) Copyright 1998 Sokkia May-29-80 23:39 01/29/1999
                Angle Degrees        Dist Feet
```

```
              Temp Farenht         Coord N-E-Elev
JOB              TRAV                 Point Id Alpha (14)
            Atmos crn No         C and R crn No
         Record elev Yes      Sea level crn No
POS TP 1       North 10050.000      East 10000.000        Elev 0.000
POS TP 2       North 10000.000      East 10000.000        Elev 0.000
POS TP 3       North 9515.636       East 9551.975         Elev 37.611
              Code T3
POS TP 403   North 4967.527       East 5074.632         Elev 0.000
NOTE TS        Jan-01-80 00:14
** End of report **
```

Sokkia G2 This routine takes you directly to the SurvCOM dialog, similar to the Carlson SurvCE process.

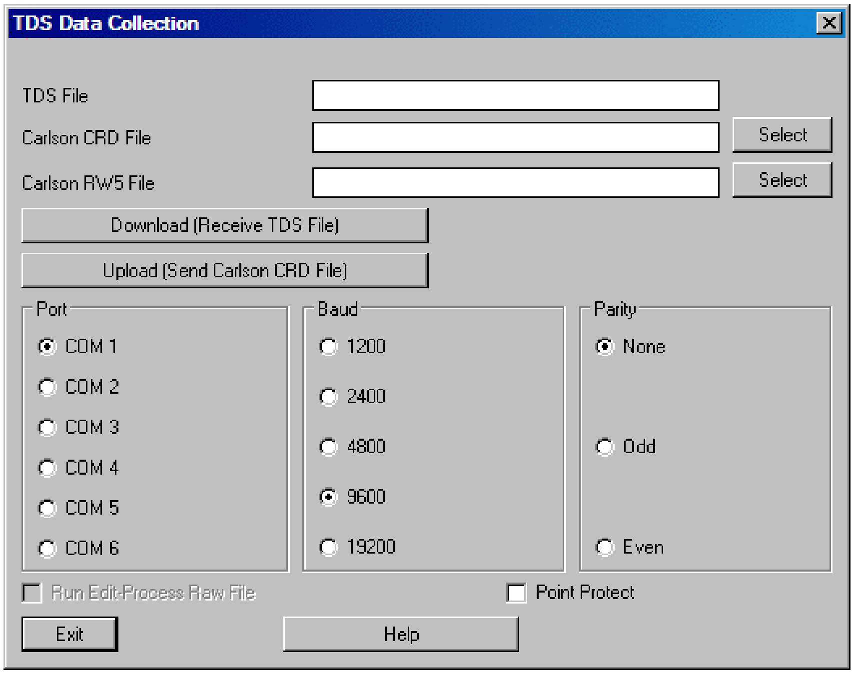
TDS

Download [HP-48 and Husky]

In the TDS program, go to the File Transfer routine. Choose the type of data to transfer (CRD or RAW). Next pick the Send function key. Stop here on the TDS and go to Carlson to run *Data Collection* in the Survey menu and pick TDS. Make sure that the COM port and baud rate are set correctly. Then pick the Download button. The Carlson program will now wait to receive the TDS file. Within 10 seconds select the file to send on the TDS. The file should be transferred now. When the transfer is complete, the program will ask you for the Carlson file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Download [Ranger and Windows CE]

In the TDS program, go to the Transfer routine and pick the Send File function. Set the "Connecting To" field to HP-48. Make sure that the COM port, baud rate and parity are set correctly and then pick OK. In the Type field of the file selection dialog, choose Coordinate Files or Raw Files. Stop here on the TDS and go to Carlson to run *Data Collection* in the Survey menu and pick TDS. Make sure that the COM port and baud rate are set correctly. Then pick the Download button. The Carlson program will now wait to receive the TDS file. Within 10 seconds select the file to send on the TDS and pick OK in the TDS dialog. The file should be transferred now. When the transfer is complete, the program will ask you for the Carlson file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.



Upload [HP-48 and Husky]

A Carlson coordinate (.CRD) file can be converted to a CR5 file and uploaded into TDS. Start in the TDS program, by going to the File Transfer routine. Then move back to Carlson and run *Data Collection* in the Survey menu and pick TDS. In the Carlson dialog, enter a TDS File name. This name should not include the drive and directory path or file extension. For example, if the coordinate (.CRD) file is c:\scadxml\data\simo2.crd then the TDS File name could be just SIMO2. Next pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Now pick the Carlson Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the range of points but before clicking the Start Transfer button go to TDS and hit the Receive function key. Within 10 seconds go back and click the OK button on the range of points. The file should then transfer.

Upload [Ranger and Windows CE]

A Carlson coordinate (.CRD) file can be converted to a CR5 file and uploaded into TDS. Start in the TDS program, by going to the Transfer routine and pick the Receive File function. Set the "Connecting To" field to HP-48. Make sure that the COM port, baud rate and parity are set correctly and then pick OK. Then move back to Carlson and run *Data Collection* in the Survey menu and pick TDS. In the Carlson dialog, enter a TDS File name. This name should not include the drive and directory path or file extension. For example, if the coordinate (.CRD) file is c:\scadxml\data\simo2.crd then the TDS File name could be just SIMO2. Next pick the Select File button next to the Carlson coordinate (.CRD) file edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Now pick the Carlson Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the range of points and click the Start Transfer button.

SMI

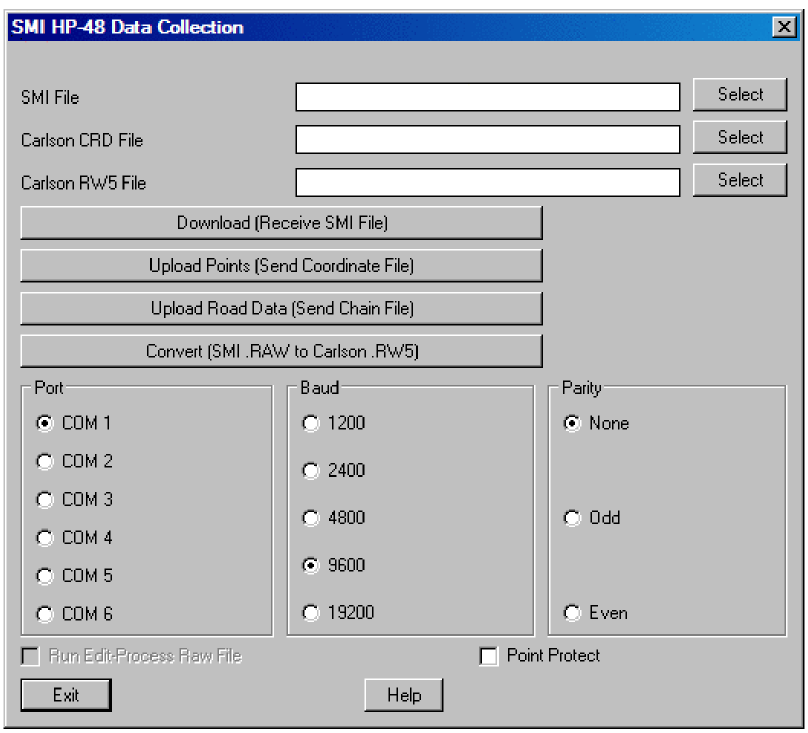
Download

To send point data from the SMI data collector, go to the file transfer routine by typing [More] [NXT] [TOPC] [COMM]. In SMI version 6 or later, type [Job][KERM][SEND]. Also in version 6, make sure that the first function key reads [NE] and not [XY] in the [Job][KERM] screen. Otherwise the coordinate northing and easting will be reversed. The [NE] stands for North-East coordinate order which is the format that Carlson expects. Also in the

[Job][KERM] screen, make sure that the second function key reads [COMM] and not [SPACE]. The [COMM] stands for comma separators. Then enter the first point to send followed by the last point to send but before pressing Enter for the last point go to Carlson. Run *Data Collection* in the Survey menu and choose SMI. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to SMI and press Enter for the last point to send. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector. To send raw data, use the [Print][Raw] routine in SMI along with the same Carlson procedure used for point data.

Upload

From the SMI data collector, go to the file transfer routine by typing [More] [NXT] [TO48] [COMM]. In SMI version 6 or later, type [Job][KERM][RECV]. Also in version 6, make sure that first function key reads [NE] and not [XY] in the [Job][KERM] screen. Otherwise the coordinate northing and easting will be reversed. Then enter the first point to send followed by the last point to send. Next enter the job name but before pressing Enter go to Carlson and run SMI under *Data Collection* in the Survey menu. In the Carlson dialog, specify the same job name as entered in SMI. Next pick the Select File button next to the Carlson CRD File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the same range of points as entered on the SMI. Go back to SMI and hit Enter for job name followed by clicking the OK button for range of points in Carlson. The file transfer should now go.



Leica

There are two types of Leica transfers: GIF-10 and GeoCom for all other Leica instruments. The type is set in the Equipment Type field on the main dialog. For transferring with the Leica instruments, the GeoCom program shows a dialog of the available COM ports on your computer. On the first time that you transfer to an instrument, you will

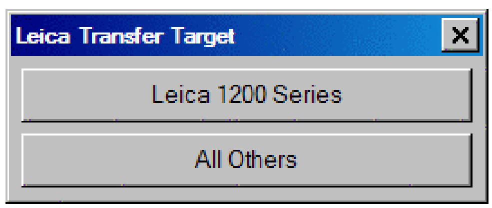
need to pick the Instruments button and register the instrument from the list. Pick the Port Settings button to make sure that the communication settings match the instrument.

To download a file with GeoCom, make sure that the instrument is ON and connected to the computer by serial cable. The instrument also needs to be in GeoCom mode. Then pick the Download in the Carlson dialog. In the GeoCom program, open the computer COM port that the instrument is connected to by picking the '+'. Then open the Memory Card and GSI folders. Next select the file to transfer and click the OK button. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

To upload a file with GeoCom, specify the file name to be created on the instrument in the Leica File field and pick the Upload button in the Carlson dialog. Then the program will prompt for the range of points to transfer. Fill out the range and pick the Start Transfer button. Then the GeoCom program will start. Open the computer COM port by picking the '+'. Then open the Memory Card folder and highlight the GSI folder and click OK.

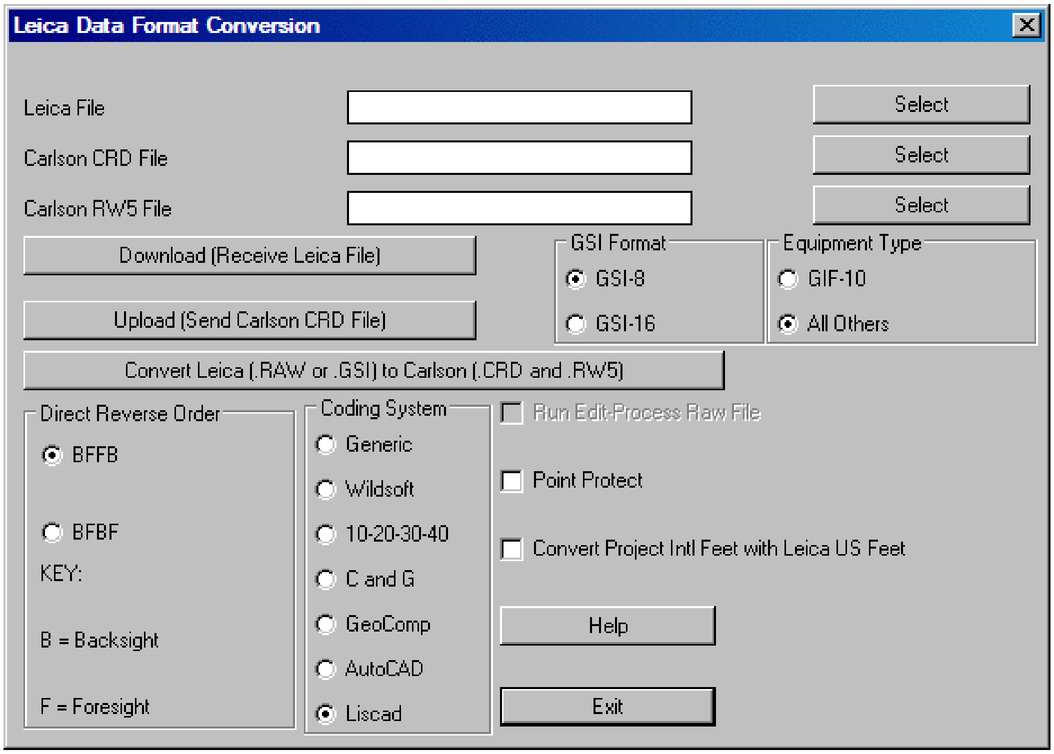
The upload and download file transfer works with the GIF-10 data collector. The GIF-10 communication settings should be the following:

Baud: 9600
Parity: NONE
Protocol: NONE
Stop Bit: 1
End Mark: CR/LF
Connected As: Some computers use DCE and others use DTE



First Leica dialog

When Leica 1200 Series is chosen



All others

Download
From the GIF-10, go to the file transfer routine. Then go to Carlson and run *Data Collection* in the Survey menu and choose Leica. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to GIF-10 and select the file to send. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload
From the GIF-10 data collector, go to the file transfer routine. Then go to Carlson and run Leica under *Data Collection* in the Survey menu. In the Carlson dialog, specify the job name in the Leica File field. Next pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Before clicking the OK button for range of points, go to GIF-10 and start the receive by highlighting Receive and pressing the Run button. The GIF-10 now shows the available job numbers. Choose a job to receive the transfer using the arrow buttons and then press the Run button.

Converting
Carlson supports raw and coordinate data collected using three different Leica Operation Codes: Wildsoft and 10-20-30-40 as well as the newer LISCAD. Moreover, data could be in the GSI8 format or the newer GSI16 format. Some example files are shown here.

GSI8 format data file using LISCAD Operation codes:
WILD GIF-12
410149+00000001 42....+00005003 43....+00005.42 44....+00005.25 45....+00005000
110150+00005000 21.324+35959480 22.324+09238590 31..01+00228271

410151+00000005 42....+00010100
110152+00005001 21.324+35156390 22.324+09303500 31..01+00133532
410153+00000005 42....+00070100
410154+00000014 42....+00000ELM
110155+00007082 21.324+34739450 22.324+09322050 31..01+00137685
410156+00000005 42....+00070102

GSI16 format data file using LISCAD Operation codes:

∗110001+0000000000000001 84..11+0000010000000000 85..11+0000003000000000
86..11+0000000001000000 87..11+0000000000005170
∗410002+0000000000000009 42....+0000000000000001 43....+000010000000.000 44....+000003000000.000
45....+000000001000.000
∗410003+0000000000000001 42....+0000000000000001 43....+000000000005.330 44....+000000000000.000
∗410004+0000000000000004 42....+00000000178.1530
∗410005+0000000000000003 42....+0000000000000002 43....+0000000000000001
∗110006+00000000000000RO 21.324+0000000017815300 22.324+0000000008424260
31..01+0000000000000000
∗410007+0000000000000100
∗410008+0000000000000012 42....+000000000005.090
∗110009+0000000000000002 21.324+0000000000831230 22.324+0000000008130270
31..01+0000000000089996
∗110010+0000000000000002 21.324+0000000018831230 22.324+0000000027829250
31..01+0000000000089996
∗110011+00000000000000RO 21.324+0000000035815170 22.324+0000000027539300
31..01+0000000000000000

GSI8 format data file using Wildsoft Operation codes:

410001+00000001 42....+00000013 43....+00000000 44....+00000012 45....+00981101
410002+00000002 42....+00000013 43....+00005.42 44....+00000012 45....+00000000
410003+00000032 42....+00000500 43....+00004.26 44....+00000012 45....+00000000
410004+000000TP 42....+00000000 43....+00000000 44....+00000000 45....+00000000
110005+00000501 21.124+00000000 22.104+09136260 31...1+00000000 51..0.+0012+000
110006+00000502 21.124+03741320 22.104+08915570 31...1+00246818 51..0.+0012+000
110007+00000503 21.124+03915180 22.104+08919040 31...1+00251956 51..0.+0012+000
110008+00000504 21.124+06530420 22.104+08839360 31...1+00113998 51..0.+0012+000

Leica raw files usually have a .RAW or .GSI extension. The primary difference in the GSI8 and GSI16 formats is that information is contained in data blocks of 16 characters in the GSI16 format, while it is contained in blocks of 8 characters in the GSI8 format. Leica instruments make it possible to have both the GSI8 as well as GSI16 data formats in the same raw file. However, lines with the GSI16 format data will always start with an asterisk (∗) character, to distinguish them from the GSI8 format. There is no distinction between Leica raw files collected in the Wildsoft and LISCAD operation codes.

Supported LISCAD codes:

1: New instrument setup
2: New target height
3: Sets of directions
4: Fixed azimuth
5: Feature code
6: Measured offset
8: Line creation for sub-codes 1 (straight string), 2 (curved string) and 6 (arc by 3 points)

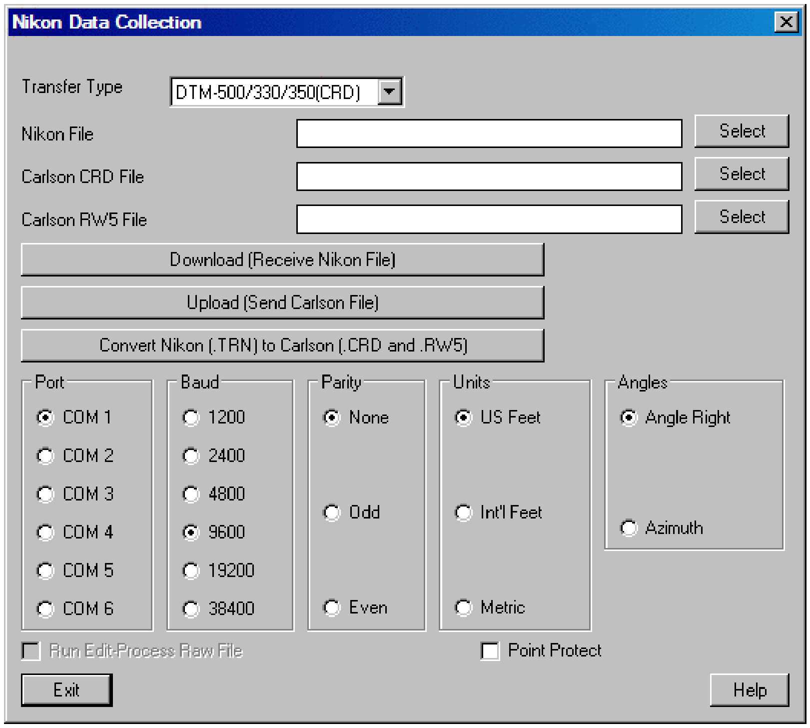
9: Fixed coordinates
11: Close string
14: Additional description
20: Start of job
27: Feature code
90: Split feature code
100+: Descriptions

The Convert button can be used to convert any Leica format file into a Carlson format file. For example, if you have a Leica PCMCIA card then there is no serial cable transfer to do. Instead use the Convert routine to make the Carlson raw data (.RW5) and coordinate (.CRD) files. Since there is no distinction between Wildsoft and LISCAD files, the user must know in advance which format has been used in the file. Then, select that particular option (Wildsoft, 10-20-30-40 or LISCAD) under the "Coding System" option at the bottom of the dialog box, as shown in the previous page. Another option that the user needs to choose is the order in which foresight-backsight readings have been recorded in the raw file, BFFB or BFBF, as explained in the dialog box. Then, the user can simply pick the "Convert" button and the program prompts for the input" Wild/Leica File" (raw file), and the output" Carlson RW5 file" and "Carlson CRD file", if they are not already filled.

Nikon

Download
First choose the equipment and data type under the Transfer Type list. Also check that the communication and data format settings match your collector. Then click the Download button and follow the on-screen directions. When the transfer is complete, the program will ask you for the Carlson coordinate file (.CRD) and raw file (.RW5) to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector. The original data from the collector is stored in a file name with the same name as the coordinate file except with a .TRN extension. For example, job5.crd would have job5.trn.

Upload
Pick the Select File button next to the Carlson CRD File edit box and choose the CRD file to send. Check that the COM port and baud rate are set correctly and then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Set the points and then click the Start Transfer button. The file transfer should now go.



Convert Nikon to Carlson

The Convert button will translate the Nikon raw file format (.TRN or .RAW) into Carlson coordinate (.CRD) and raw (.RW5) files.

Portion of typical Nikon file format:

MP,1,,5000.0000,5000.0000,0.0000,T/1
CO,31-Oct-1999 11:42:38
ST,1,,2,,0.0000,0.00000,0.00000
SS,3,0.0000,152.1510,359.59590,90.44100,11:43:38,T/2
SS,4,0.0000,127.5560,0.06040,90.40110,11:44:45,CON
SS,5,0.0000,97.1820,2.19580,90.52460,11:45:43,CON

Geodimeter

Download

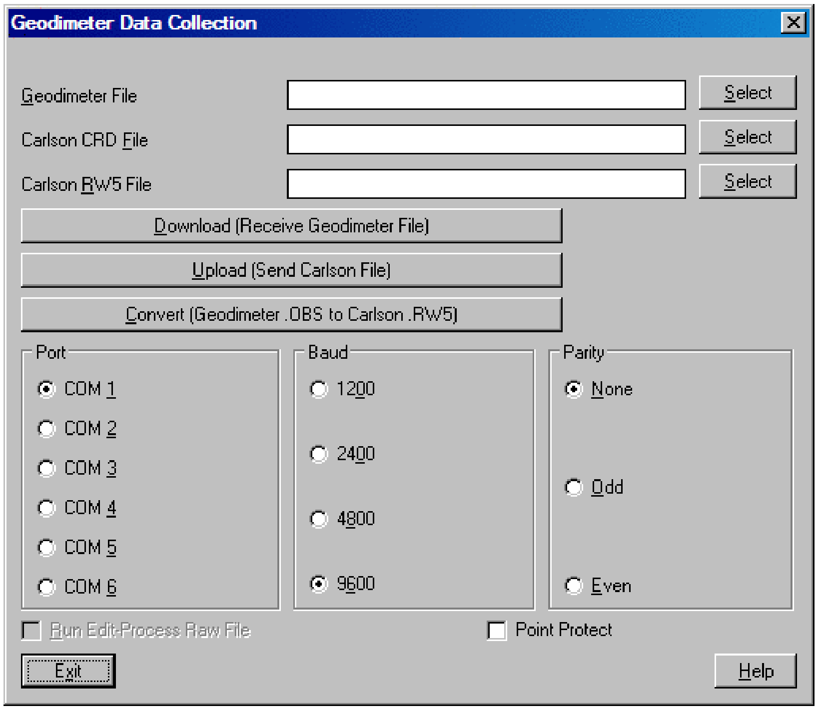
From the Geodimeter data collector, go to the file transfer routine by pressing the PRG (Program) key and entering program 54. Then choose Imem (option 1) as the source. Next choose the file type to send as either Job (measurement data) or Area (point data). The Geodimeter will then prompt for the job name. Next enter Serial (option 3) as the destination. A confirmation screen appears showing the serial port settings. Here are some typical settings:

COM=1,8,0,9600

Before pressing enter (ENT key), go to Carlson and run *Data Collection* in the Survey menu and choose Geodimeter. Then click the Download button and within 15 seconds, go back to the Geodimeter and press Enter. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate file and raw file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

In Carlson, run Geodimeter under *Data Collection* in the Survey menu. Pick the Select File button next to the Carlson CRD File edit box and choose the CRD file to send. Check that the COM port and baud rate are set correctly and then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the points to send but before clicking OK, go to the Geodimeter data collector. Start the file transfer routine by pressing the PRG key and entering program 54. Then choose Serial (option 3) as the source. The Geodimeter will display the serial port settings. Check these values and press enter. Next choose Area (option 2) as the destination. Then enter the job name. The Geodimeter is now listening for data. Quickly go back to Carlson and click OK on the points to send dialog. The file transfer should now go



Convert

The Convert button will translate the Geodimeter raw file format (.OBS) into Carlson coordinate (.CRD) and raw (.RW5) files.

Communication Settings

If the Geodimeter is not communicating with Carlson, run function 79 on the Geodimeter and make sure that it is set to 4. This setting is for the transfer message end of sequence format.

Supported Geodimeter Codes

The following Geodimeter codes are processed when converting the Geodimeter raw file. All other codes are recorded as descriptions in the Carlson rw5 file.

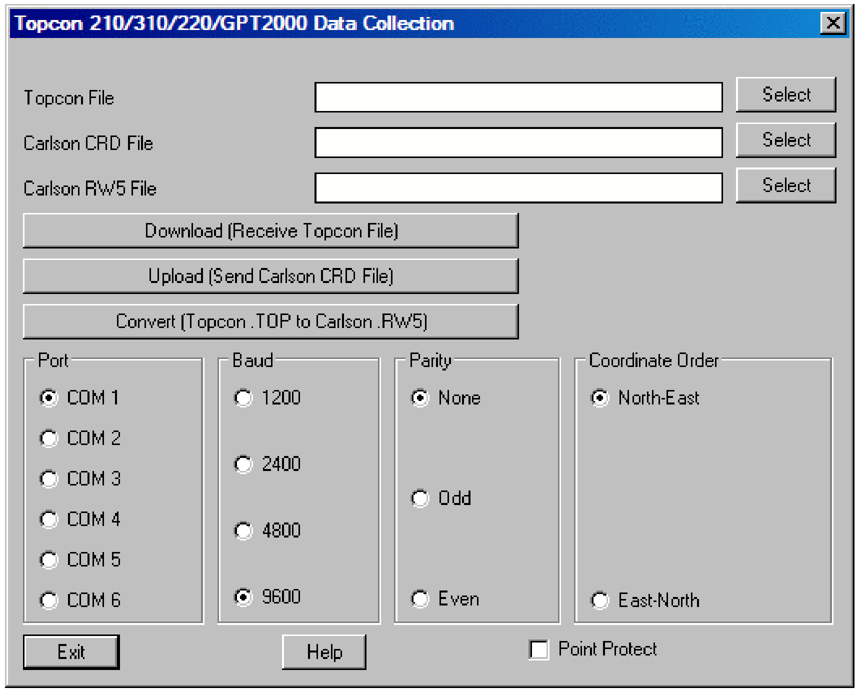
0=Info
1=Data
2=Station No
3=Instrument Height
4=Point Code
5=Point Number

6=Signal Height
7=Horizontal Angle
8=Vertical Angle
9=Slope Distance
11=Horizontal Distance
17=Horizontal Angle
18=Vertical Angle
21=Horizontal Reference Angle
30=Atmospheric Correction
37=Northing
38=Easting
39=Elevation
40=Delta North
41=Delta East
42=Delta Elevation
45=Correction To Bearing
46=Standard Deviation
50=Job Number
51=Date
52=Time
53=Operator
54=Project Id
55=Instrument Id
56=Temperature
60=Shot Id
61=Activity Code
62=Reference Object
70=Entered Radial Offset
71=Entered Angle Offset
72=Calculated Radial Offset
73=Calculated Angle Offset
74=Air Pressure

Portion of typical Geodimeter file format
5=108
4=13POC
6=5.000
7=238.0708
8=89.2236
9=440.39
37=767.42
38=4626.07
39=699.795

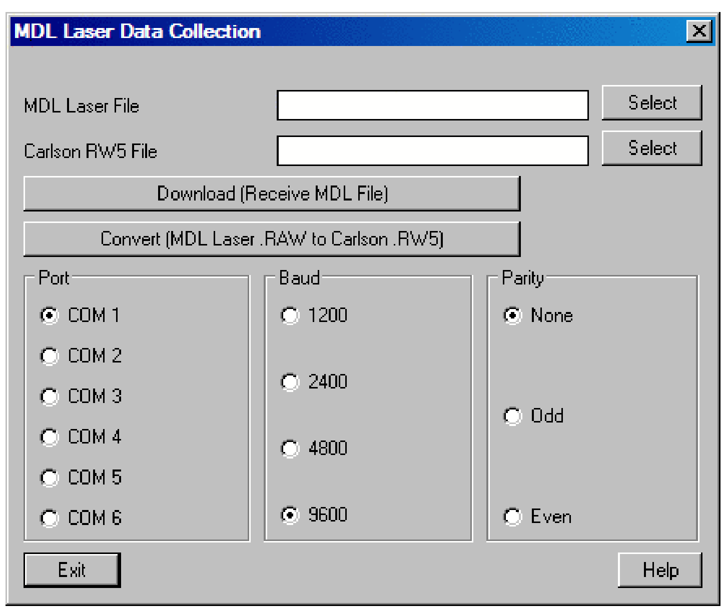
Topcon 210/310/220/GPT2000

This command supports these above Topcon models.



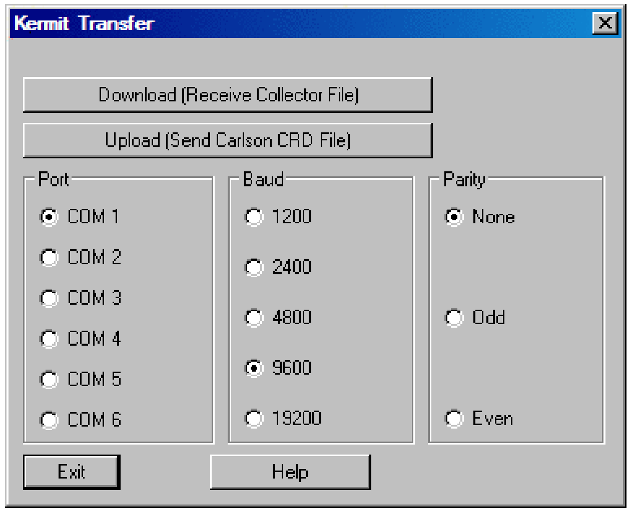
MDL Laser

The MDL Laser outputs a raw file of angles, distances and codes as one long string of data which can be converted into a Carlson raw data (.RW5) file. There is no coordinate data in the MDL raw file. So you need to run *Edit-Process Raw File* to calculate coordinates from the raw data. The Download button will transfer the MDL raw data from a BDI logger.



Kermit

Kermit can be also used for transfering files with accuracy. The dialog looks like this:



Keyboard Command: datacolt
Prerequisite: None

Convert LDD-AEC Contours

Function

This command allows you to convert LandDesktop contours (known as AECC_CONTOUR objects) into polylines. You must have the AEC Object Enabler installed before using this command. If you do not have the object enabler installed, download the latest version from www.autodesk.com.

You can use the *List* command to determine if contours are polylines or AECC_Contour objects. Here is an example listing:

AECC_CONTOUR Layer: "CONT-MJR"

Space: Model space

Handle = 429

Major Contour Interval

Elevation: 1005.00

Smoothing: None

Number of Vertices: 48

Open

Length: 560.25

Constant width: 0.00

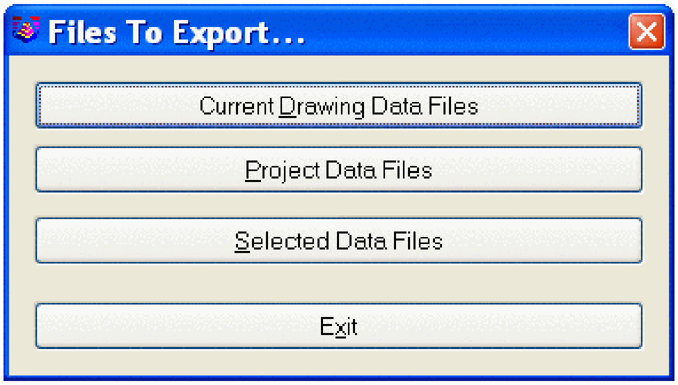
Style Name: Standard

Import/Export LandXML Files

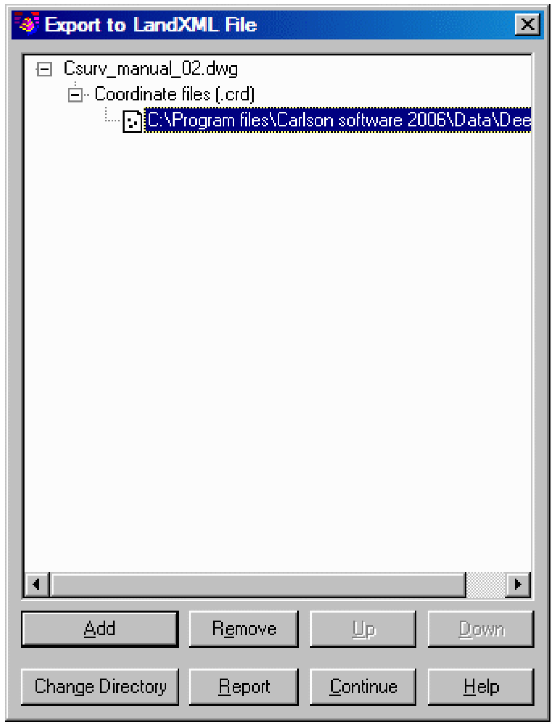
This command performs two functions: Export Carlson files to LandXML and Import LandXML files into Carlson. This command supports version 1.0 of LandXML and the following Carlson file types: Coordinate (.CRD) files, Centerline (.CL) files, Profile (.PRO) files, Section (.SCT) files, Grid (.GRD) files, Triangulation (.FLT) files, and Lot (.LOT) files.

Export to LandXML: Allows you to export a Carlson file to LandXML. Choose a new or existing LandXML file, then choose an existing Carlson file.
Import to Carlson: Allows you to import a LandXML file into Carlson. First, choose a new or existing Carlson file, then choose an existing LandXML file.

You first see the Import/Export LandXML dialog. To Export, for example, first click Export to LandXML. Then click Current Drawing Data Files button. You will then see the Select LandXML File dialog box. Select an existing .XML file to be exported from the list of files on the right and click Open.



The next dialog appears, showing the new buttons to be used for the Export. Choose the Add, Remove, Change Directory and/or Report buttons, and then Continue.



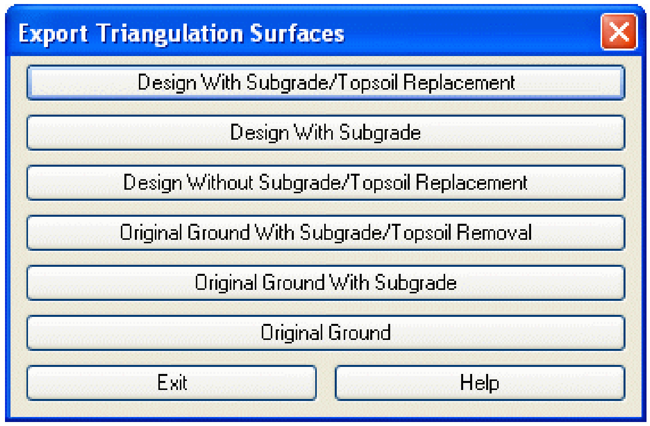
Choosing the Continue button takes you further into the Export process. The Report button will give you the Report Formatter Options dialog box.

Data protection is turned on by default, meaning that if you are importing/exporting to an existing file, you will be prompted before the program overwrites existing data. There is a Point Protection option that will not allow any points to be overwritten if the same number appears more than once in the XML file being imported. If you decide to Import From LandXML, go back to the opening dialog an choose the existing LandXML file.

Pulldown Menu Location: File
Keyboard Command: imp_exp
Prerequisite: Files to convert
File Name: \lsp\gisutil.arx

Import/Export Carlson Triangulation Files

Import Carlson Triangulation Files allows you to import an external surface file into TakeOff to use as a named surface. Export Carlson Triangulation Files allows you to take a Surface Triangulation file and save it independent of the drawing.



Prerequisite: .TIN or .FLT files

Keyboard Command: import_tin, export_tin

Import/Export DXF Files

Import DXF File allows you to import an external DXF file into the Carlson TakeOff drawing (DWG). Export DXF File exports selected entities from the Carlson TakeOff drawing (DWG) to a DXF file. The DXF file is another format for the drawing objects of the DWG file. Typically, other CAD programs at least support DXF files if they don't support DWG. So the DXF file (Drawing eXchange File) can be used to transfer drawing data between Carlson Takeoff and other non-DWG programs.

Prerequisite: a DXF file

Keyboard Command: _dxfin, _dxfout

Import Polyline File

Function

This command draws polylines from the selected polyline file. These polylines are drawn in the current layer. This command supports the following formats: Carlson (.PLN), Idan (.DIS), MicroStation (.TXT), MOSS (.INP, .PRN) and Topcon Pocket 3D (.TXT).

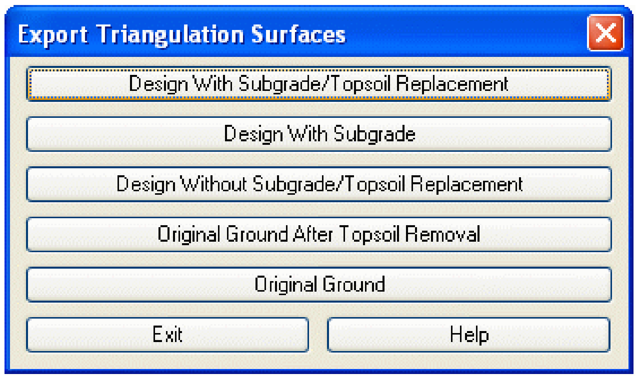
Prompts

Polyline file format [<Carlson>/DTM/Idan/MicroStation/MOSS/Topcon]? *press Enter to accept Carlson*
Polyline File to Read Dialog *select existing .PLN file*

Keyboard Command: polydraw
Prerequisite: A polyline file

Export Surface DXF Files

Export Carlson Triangulation Files allows you to take a Surface file and save it as a DXF file. The DXF file will contain 3D Faces for the triangulation surface. This command is another way besides the LandXML routine to transfer a Carlson Takeoff surface to other programs.



Prerequisite: .TIN or .FLT files

Keyboard Command: EXPORT_DXF

Export Polyline File

This command creates a polyline file that contains the point data of the select polylines. The objects supported by this tool include polylines, arcs and lines. If you want to include text, you must use the *Text Explode To Polylines* command found in the Edit menu to convert the text to polylines before running this command. This polyline file is a text file that has three formats. The Carlson format (.PLN) is used by machine control (Carlson Grade, Dozer 2000, GradeStar) for the plan view. Each polyline begins with a line of "POLYLINE, Color number". Then the points for the polyline are listed on separate lines in X,Y,Z format. Here is a list of the available color numbers:

0 = Black
1 = Blue
2 = Green
3 = Cyan
4 = Red
5 = Magenta
6 = Brown
7 = Light Gray
8 = Dark Gray
9 = Light Blue
10 = Light Green
11 = Light Cyan
12 = Light Red
13 = Light Magenta
14 = Yellow
15 = White

The MicroStation format (.txt) can be imported into MicroStation. This format has the coordinates as space

delimited for each polyline point. There is an extra column with a 1 or 0 where 1 specifies the start of a new polyline. The DTM and Idan formats create linework files for the DTM and Idan programs.

Prompts

Polyline file format [<Carlson>/DTM/Idan/MicroStation]? *press Enter for Carlson format*
Specify File to Write dialog *create a new file or append to existing*
Polyline file for Grid File Utilities macro [Yes/<No>]? *press Enter* The option will write a polyline file that can be used with Grid File Utilities for inclusion/exclusion perimeters.
Include Z coordinate in polyline file [Yes/<No>]? *press Enter* This option controls whether the polyline vertices are written in 2D or 3D.
Specify Exclusion/Warning Polylines [Yes/<No>]? *press Enter* This option applies to machine control for warning areas.
Specify WorkZone Polylines [Yes/<No>]? *press Enter* This option applies to machine control for working areas.
Reduce Polyline Vertices [<Yes>/No]? *press Enter* This option applies Reduce Polyline to the polyline vertices before writing the file.
Enter reduce offset cutoff <0.1>: *press Enter*
Decimal places for coordinates <2>: *press Enter*
Select polylines, lines and arcs to write.
Select objects: *pick the entities to process*
Done.

Sample Polyline File:

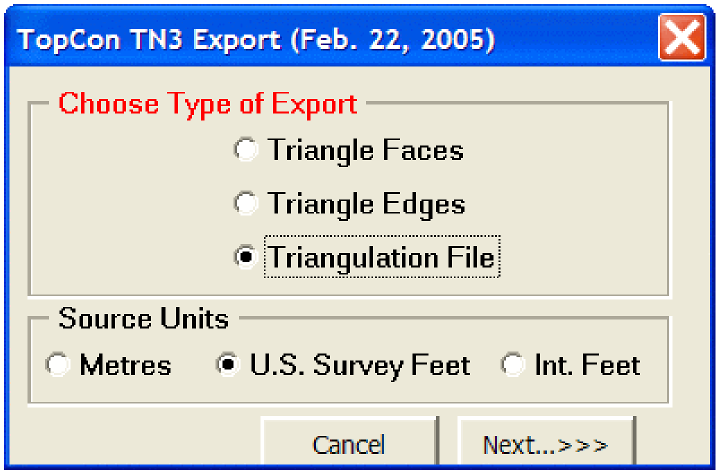
POLYLINE,15
47639.82,74540.11,0.00
47670.49,74565.79,0.00
47701.08,74591.49,0.00
49375.61,76358.47,0.00
50066.86,76846.75,0.00
POLYLINE,15
47633.24,74547.97,0.00
47663.90,74573.65,0.00
etc...

Keyboard Command: polywrite
Prerequisite: Polylines in the drawing

Export Topcon TIN File

Function

This command writes a Topcon TIN file (.TN3) from a Carlson Takeoff surface. The first prompt at the Command line chooses whether to export the surface from a file or the screen. The file option will prompt for a triangulation file (.tin or .flt) and then the Topcon TIN file to create. The screen option can be used to export a surface from 3d Faces or TIN lines from the drawing. The screen option has the following prompts:



•
Choose Type of Export

- **Triangle Faces:** Triangulation networks that consist of edge matched 3DFaces defined by three points.
- **Triangle Edges:** Triangulation Networks that consist of edges drawn with Lines defined by two points.
- **Triangulation File:** A Carlson triangulation file (with a DOS extension of .flt).

Source Units: Select the appropriate unit type of the entities to be exported, then press NEXT.

If the Triangulation File option is chosen, a standard windows file selection dialog will open to allow for the selection of the .flt file.

If Triangulation Edges or Faces are used for the export, you must select the entities to be exported from the screen. The following dialog opens:

Manually Select allows on-screen selection with any of the various methods (Window, Crossing, Crossing Polygon,

etc.)

All on a Selected Layer allows selection of a single entity. The command then selects all Lines or 3DFaces on the layer of the selected entity for export, filtering entities not on selected layer.

Generally, you can select any entities drawn on the screen without special care. The command filters out all types except Lines or 3DFaces depending on the type selected.

When Processing Lines: Crossings, crossing polygon, and fence selection sets can often result in "spikes" around the edges. The command generally discards these spikes, or reconnects them. In any case, better results are developed with clean selections of drawing entities.

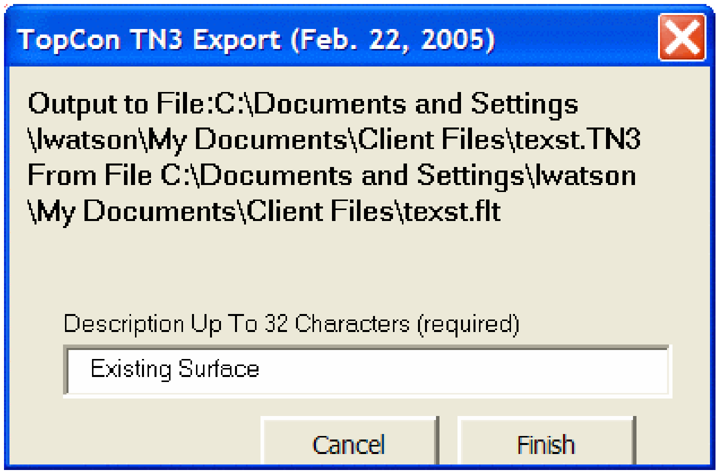
Pressing NEXT will open the following dialog:

Other Features to Export: Often times, landmark lines, or other features will assist in orientation to the user when using the TN3 file in the field. For example: Property lines, project centerlines, or other well established reference lines. These features may be added and displayed on the TN3 by choosing them at this prompting. The Other Features dialog opens with all three Source Formats.

Selections are made as described above.

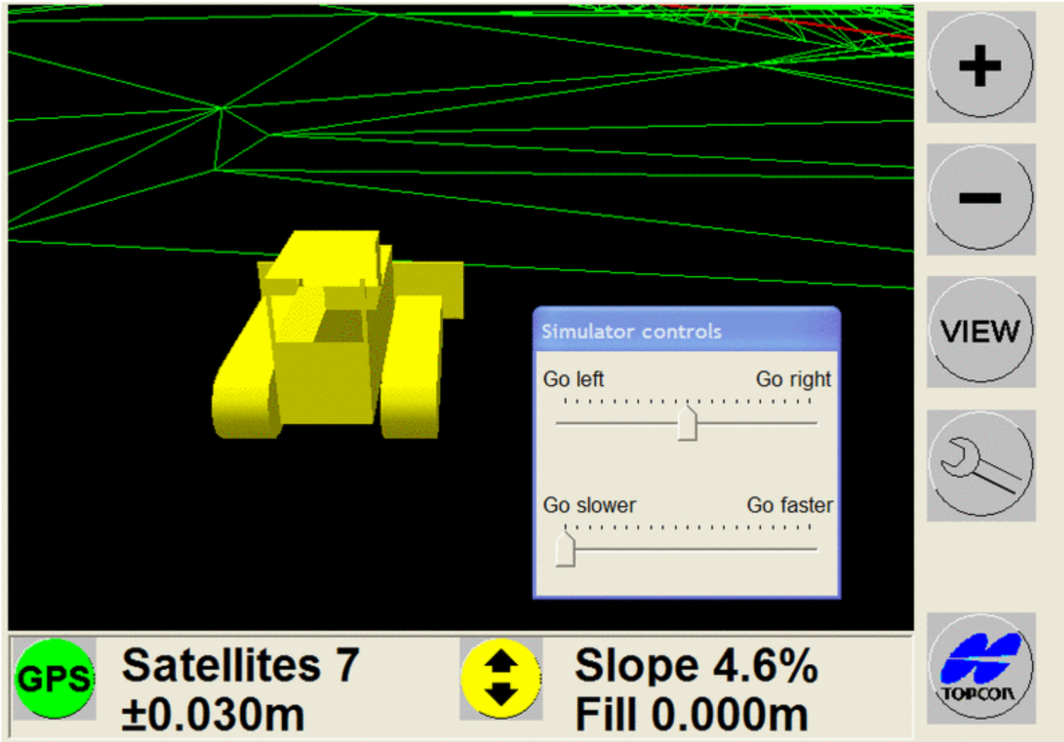
Once the selections are made and the dialog reappears, press NEXT.

Naming the TN3 File: After all the triangulation and feature items are selected from the graphics screen, the command requests a name for the output file. Once a name is given, the following dialog reviews all the selections made. A description for the job can be added at this time. Press Finish when ready to proceed.



The program will process the selected entities and/or files and report as shown here:

To view the final output, press the Run Simulation button, or press Exit to return to the Carlson program.



Run Simulation Dialog

Clipboard

This command allows for different cut, copy, and paste options.

Cut

To cut objects to the Clipboard.

- Select the objects you want to cut.
- From the Clipboard command, choose Cut.

The objects are available to be pasted into other Windows applications.

Copy

To copy objects to the Clipboard.

- Select the objects you want to copy.
- From the Clipboard command, choose Copy.

Copy with Base Point

To copy objects to the Clipboard. When the objects are pasted into a drawing, the program places them relative to the specified base point.

- Select the objects you want to copy.
- From the Clipboard command, choose Copy with Base Point.
- Specify the base point.

Paste

The objects currently on the Clipboard are pasted into the drawing at the specified insertion point.

- From Clipboard command, choose Paste.

Paste as Block

The objects currently on the Clipboard are pasted into the drawing as a block at the specified insertion point.

- From Clipboard command, choose Paste as Block.

Paste to Original Coordinates

The objects currently on the Clipboard are pasted into the drawing using the coordinates from the original drawing.

- From Clipboard command, choose Paste to Original Coordinates.

Display Last Report

Function

This brings up the last report generated by any Carlson Takeoff command that uses the standard report viewer.

Prerequisite: a previously generated report

Keyboard Command: last_report

Edit Menu 3

Undo

This command allows you to reverse the effect of previously issued commands.

Prerequisite: None

Keyboard Command: U

Redo

This command allows you to reverse the effects of the previous UNDO command.

Prerequisite: None

Keyboard Command: REDO

Erase Select

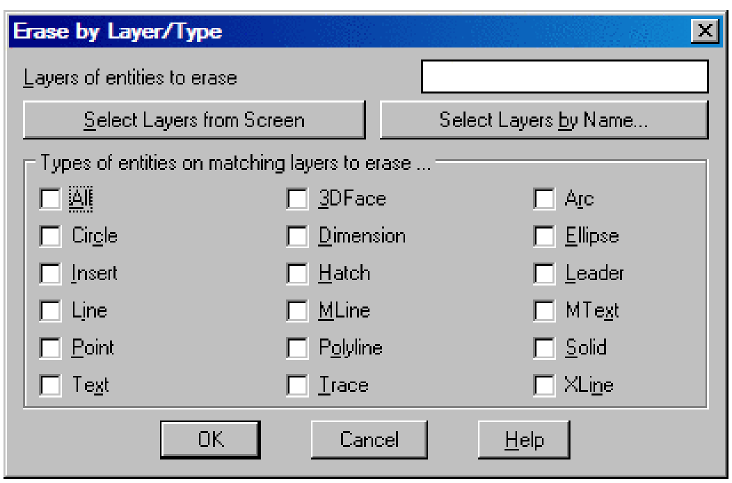
This command allows you to remove objects from a drawing.

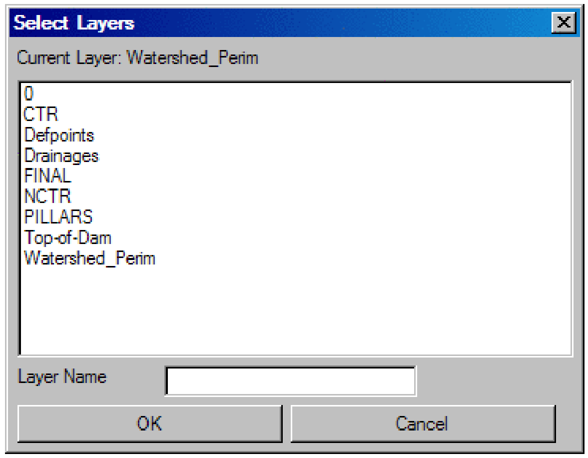
Prerequisite: None

Keyboard Command: ERASE, E

Erase by Layer

This command will ERASE all the entities on the specified layers but will not delete these layers from the drawing. The command prompts for the layer name to erase and then erases all entities on that layer. In addition to typing in the layer name, you can also specify a layer to delete by picking an entity on that layer. To select layers by picking, first click the **Select Layers from Screen** button and then select the entities on the layers to be deleted. The **Select Layers by Name** button allows you to choose a layer name from a list of layers in the drawing. You can also specify which types of entities to erase. For instance, if you have both linework and points on the same layer and you want to erase only the linework, you can click off All and check Line and Polyline.



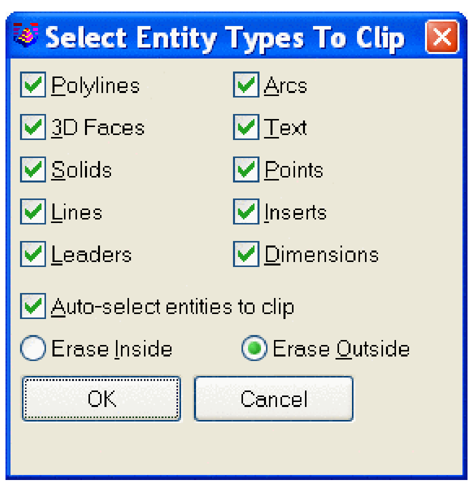


Pulldown Menu Location: Edit > Erase
Keyboard Command: ldel
Prerequisite: Something to erase
File Name: \lsp\dellayr.lsp

Erase by Closed Polyline

This tool is used to clean up drawing geometry at the extents of a polyline boundary. It provides options to erase adjacent geometry as well as trim geometry crossing the fence of the polyline.

First, select the boundary polyline. Only one can be selected. Designate the desired options in the following dialog. The top section of the dialog allows you to toggle which object types should be affected by the operation. Note that some of the objects, such as text and inserts, cannot be trimmed.



In the middle of the dialog is a toggle that determines whether to prompt for objects to process. If you want to isolate the drawings contents to that of the selected polyline, turn this toggle on. Note that all geometry in the drawing is effected, even geometry that is outside of the current viewport. Many users will prefer to turn this toggle off, so that they can be prompted to manipulate the geometry.

The bottom row allows you to choose whether to erase all the entities on the inside or outside of the polyline.

Pulldown Menu Location: Edit > Erase
Keyboard Command: erasepline
Prerequisite: Entities and a closed polyline
File Name: \lsp\poly3d.arx

Erase Outside

This command erases all the entities outside of a user specified window. This can be useful if you somehow place entities way outside your drawing limits and want to easily erase them.

Prompts

Pick 1st corner of window to erase outside of: *Pick point location*
Pick 2nd corner: *Pick second point location*

Pulldown Menu Location: Edit > Erase
Keyboard Command: eraseout
Prerequisite: Entities to erase
File Name: \lsp\surv1.lsp

Move

This command allows you to displace objects a specified distance in a specified direction.

Prerequisite: None

Keyboard Command: MOVE, M

Standard Copy

This command copies all objects you select to the Clipboard. You can paste the contents of the Clipboard into a document or drawing as an OLE object.

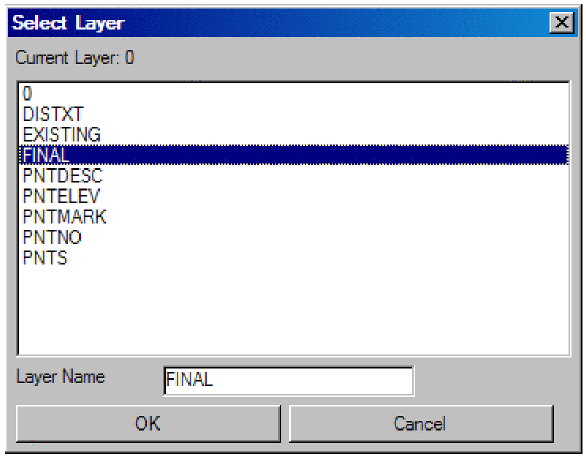
You can also use CTRL+C to run this command. If the cursor is in the drawing area, Carlson TakeOff copies the selected objects to the Clipboard. If the cursor is on the command line or in the text window, the program copies the selected text to the Clipboard.

Prerequisite: None

Keyboard Command: COPY

Copy To Layer

This command is used to copy a selected entity or entities and put the copy in a specified layer. Once copied to the chosen layer the entity or entities will take on the characteristics of that layer (color, linetype, etc.).



Prompts

Select entities to copy.
Select objects: *select entities*
Select Layer dialog *select a layer from list and click OK*

Pulldown Menu Location: Edit > Copy
Keyboard Command: copy2layer
Prerequisite: Entities to be copied
File Name: \lsp\surv1.lsp

Standard Explode

This command allows you to break a compound object into its component objects.

Results differ depending on the type of compound object you're exploding. The following is a list of objects that can be exploded and the results for each.

- **All Explodable Objects**: Produces object geometry that may look the same, but the color, linetype, and lineweight of the object may change.

- **Block**: Removes one grouping level at a time. If a block contains a polyline or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

Blocks with equal X, Y, and Z scales explode into their component objects. Blocks with unequal X, Y, and Z scales (nonuniformly scaled blocks) might explode into unexpected objects.

When nonuniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with a "∗E" prefix) and referenced with the nonuniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a nonuniformly scaled block cannot be exploded.

Exploding a block that contains attributes deletes the attribute values and redisplays the attribute definitions.

- **2D and Lightweight Polyline**: Discards any associated width or tangent information.

- **Wide Polyline**: Places the resulting lines and arcs along the center of the polyline. TakeOff discards any associated width or tangent information.

- **3D Polyline**: Explodes into line segments. Any linetype assigned to the 3D polyline is applied to each resulting line segment.

- **Text Explode to Polylines**: Explodes polylines depending on the font used for various annotations, this can make the resulting polylines more efficient in terms of vertex count.

- **Leaders**: Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), Mtext, or tolerance objects, depending on the leader.
- **Mtext**: Explodes into text entities
- **Multiline**: Explodes into lines and arcs.
- **3D Solid**: Explodes planar surfaces into regions. Nonplanar surfaces explode into bodies.
- **Region**: Explodes into lines, arcs, or splines.
- **Body**: Explodes into a single-surface body (nonplanar surfaces), regions, or curves.
- **Polyface Mesh**: Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.
- **Circle Within a Nonuniformly Scaled Block**: Explodes a circle within a nonuniformly scaled block into ellipses.
- **Arc Within a Nonuniformly Scaled Block**: Explodes an arc within a nonuniformly scaled block into elliptical arcs.

Prerequisite: None

Keyboard Command: EXPLODE, X

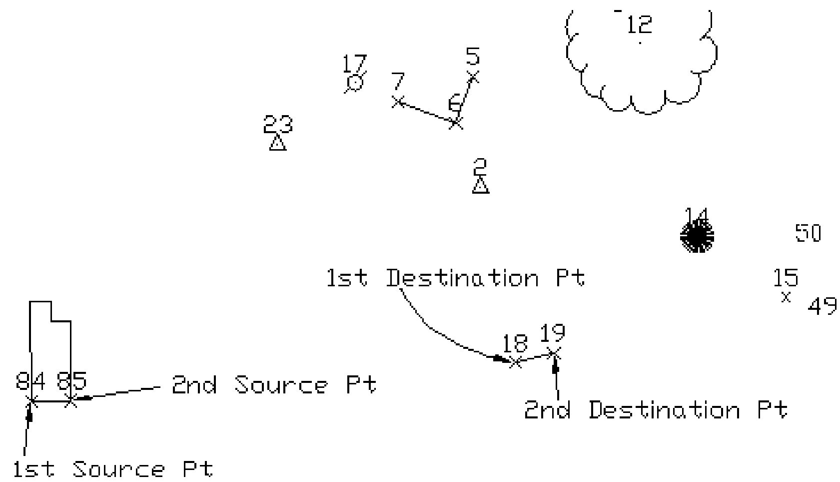
Block Explode

This command retains the values of attributes when a block is exploded. The standard AutoCAD *Explode* command changes the attribute values back to the attribute type. For example, using *Explode*, a Carlson point block would become PNTNO, PNTELEV, PNTDESC. *Block Explode* would keep the point attribute values, such as 10, 1000.0, EP. The layer names of the exploded block attributes can be either the insert layer of the parent block or the original attribute layers from the block definition.

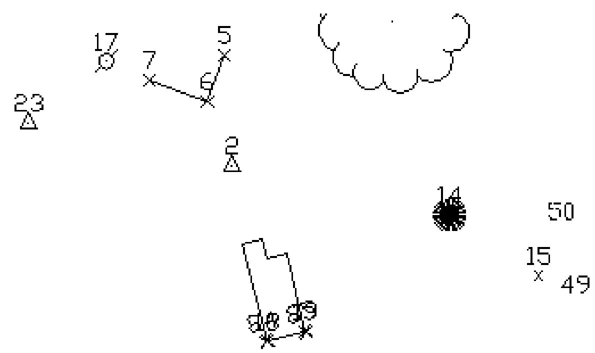
Pulldown Menu Location: Edit
Keyboard Command: explode2
Prerequisite: A block to be exploded
File Name: \lsp\blkattex.lsp

2D Align

This command will align (translate, rotate and scale) the selected objects using two pairs of source and destination control points. The difference between the first source point and first destination point determines the translation amount. The difference between the angle and distance from the first and second source points compared to the angle and distance from the first and second destination points determines the rotation and scale. The scale part of the alignment is optional. This 2D Align function is the same as the AutoCAD Align function except that this 2D Align function does not use elevations so that the alignment is always in 2D. The control points can be screen picked or entered by point numbers.



Before and after 2D Align



Prompts

Select entities to align.
Select objects: *pick entities to process*
First Source Point?
Pick point or point number: *pick point 84*
First Destination Point?
Pick point or point number: *pick point 18*
Second Source Point?
Pick point or point number: *pick point 85*
Second Destination Point?
Pick point or point number: *pick point 19*
Scale factor: 1.00434258
Scale objects based on alignment points [Yes/<No>]? *Y*
This command DOES NOT change the coordinates in the CooRDinate file!
Use Coordinate File Utilities menu, Update CRD File from Drawing.

Pulldown Menu Location: Edit > Align
Keyboard Command: scalign
Prerequisite: None
File Name: \lsp\scalign.lsp

Standard Align

Aligns objects with other objects in 2D and 3D

You use ALIGN to move, rotate, or scale objects into alignment with other objects. Add source points to the objects you want to align, and add destination points to the objects to which you want the source objects to align. You can add up to three pairs of source and destination points to align an object.

The first set of source and destination points defines the base point for the alignment. The second set of points defines the angle of rotation.

When you select three point pairs, you can move and rotate the selected objects in 3D to align with other objects.

If you use two source and destination points to perform a 3D alignment on nonperpendicular working planes, you get unpredictable results.

After you enter the points, Takeoff prompts you to scale the object. The program uses the distance between the first and second destination points as the reference length to which the object is scaled. Scaling is available only when you are aligning objects using two point pairs.

Prompts

1 Specify first source point: **pick point**

2 Specify first desitination point: **pick point**

3 Specify second source point: **pick point**

4 Specify second desitination point: **pick point**

5 Specify third source point or <continue>: **Press Enter**

6 Scale objects based on alignment points? [Yes/No] <N>: **Press Enter**

Prerequisite: None

Keyboard Command: ALIGN

Trim

This command allows you to trim objects at a cutting edge defined by other objects.

Prompts

1 Select cutting edges ...

Select objects: **pick entity**

2 Select object to trim or shift-select to extend or [Project/Edge/Undo]: **select entity to be trimmed**

- **Project**: You can project the object to be trimmed in order to trim objects that do not intersect.
- **Edge**: You can project the trimming edge in order to trim objects that do not intersect.

- **Undo**: This option allows you to undo the above projections.

Prerequisite: None

Keyboard Command: TRIM, TR

2D Scale

This command will scale the selected entities using specified scale factor and base point. This 2D Scale function is the same as the AutoCAD Scale function except that this 2D Scale function only scales the entities in the x,y coordinates and does not change the elevations of the entities. One application of this routine is to convert a drawing from architectural to decimal units when the architectural units have the drawing x,y coordinates in inches and the elevations in feet. In this case, 2D Scale can be used to apply a 1/12 scale factor (0.08333333) to convert the inches to feet for the x,y coordinates and leave the elevations unchanged.

Prompts

Select entities to scale.
Select objects: *pick entities to process*
Specify base point: *0,0*
Specify scale factor: *0.0833333333*

Pulldown Menu Location: Edit > Scale
Keyboard Command: scscale
Prerequisite: None
File Name: \lsp\surv1.lsp

Standard Scale

This command allows you to enlarge or reduce selected objects equally in the X, Y, and Z directions.

Prompts

1 Select objects: **pick entities**

2 Specify base point: pick point on screen as reference

3 Specify scale factor or [Reference]: **scale to desired size**

Prerequisite: None

Keyboard Command: SC

Extend To Edge

This command allows you to extend an object to meet another object.

Prompts

1 Select boundary edges ...

Select objects: **pick entity**

2 Select object to extend or shift-select to trim or [Project/Edge/Undo]: **pick entity**

You have the option of trimming or projecting objects and edges.

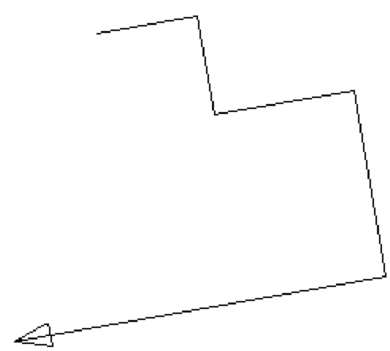
Prerequisite: None

Keyboard Command: EXTEND

Extend by Distance

This command extends a line or polyline, or creates new lines or polylines off of an existing one. By specifying a distance, a new segment of the line or polyline can be drawn from the current position. The current position and direction along the line or polyline is indicated by an arrowhead. Extend by Distance starts by selecting an existing line or polyline. Initially, the current position will be the closest vertex to where the line or polyline was selected. Extending from the endpoint of a polyline will add a new point to that polyline, while extending from any other point will create a new polyline.

There are two modes of operation: draw mode (D) and move mode (M). When in draw mode, extending will draw line or polyline segments. In move mode, the current position arrowhead can be moved without drawing segments. The orientation of the current position arrowhead can be changed with the Right, Left, and Angle commands.

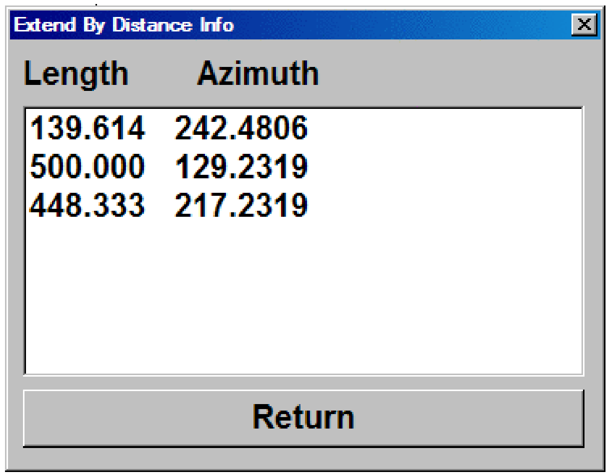


The second prompt for this command offers numerous options in the form of key letters. These key letters are listed below along with their full names and actions. The list of the Extend by Distance commands are:

 # - Number: Distance to draw or extend
A# - Angle change: Rotates pointer by specified number of degrees
A - Align: Rotates pointer to align with segment
B - Bearing: Sets pointer direction by bearing in format: Qdd.mmss with Q- quadrant, d-degrees, m-minutes, s-seconds (e.g. 130.1005 is NE 30 degrees, 10 minutes, and 5 seconds)
C - Close: Closes the polyline
D - Draw Mode: Actions draw or extend the line or polyline
E - Extend to Edge: Extends to intersection with a selected line or polyline
I - Input mode: Toggles distance input between decimal feet and feet-inches
L - Left rotate: Rotates counterclockwise 90 degrees
M - Move Mode: Actions only move the pointer
N - Next: Moves pointer forward to next point
O - Open: Opens the polyline
P - Previous: Moves pointer backward to previous point
R - Right rotate: Rotates clockwise 90 degrees
S - Switch: Reverses pointer direction

T# - Total distance: Sets current segment to specified distance
U - Undo: Undo the last Extend by Distance command
Z - Zoom mode: Toggles auto-zoom between on/off
? - Info: Displays lengths of current polyline

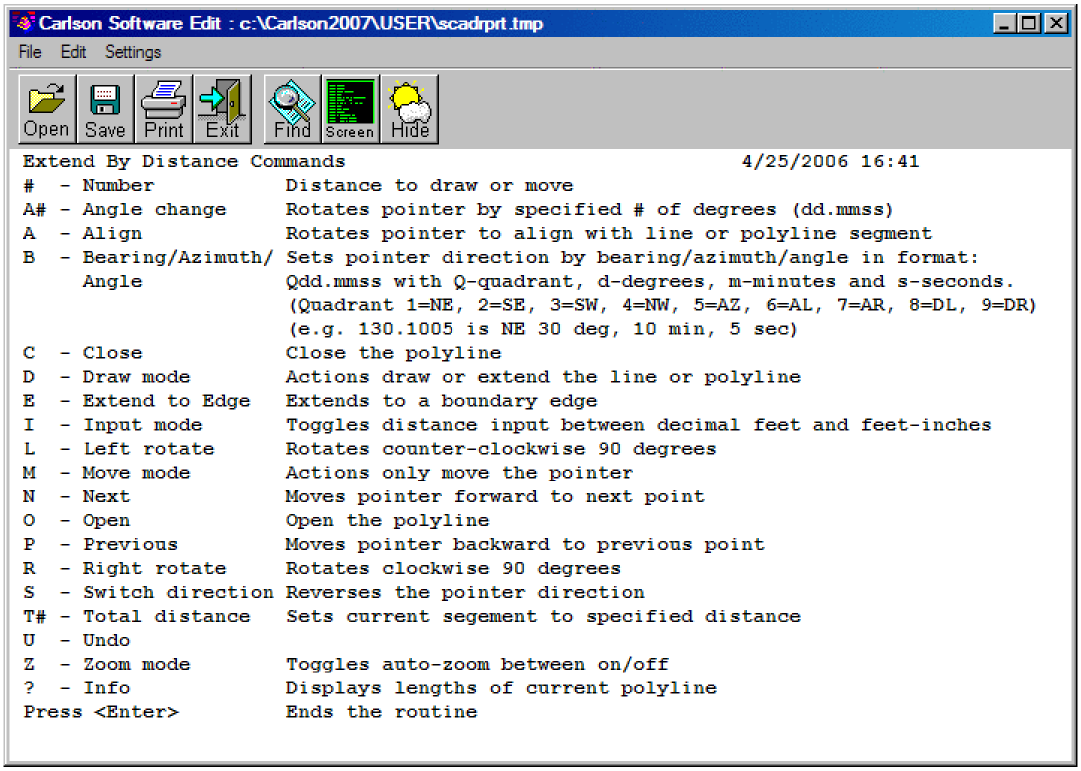
H - Help: The Help option also displays this Extend by Distance Commands list.
Press <Enter>: Ends the routine



The result of using the Info (?) feature

Prompts

Select line or polyline to extend: *select line or polyline near the place to extend*
Enter or pick distance to draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,Z,?,Help): *50* The line is extended by 50 units. Use the Pick option to pick a distance.
Pick/Horizontal Distance to Extend ([Enter] for new line): *R* Rotate right 90 degrees.
Enter or pick distance to draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,Z,?,Help): *50* The line is extended by 50 units. Use the Pick option to pick a distance.
Enter or pick distance to draw (A,B,C,E,I,L,M,N,O,P,R,S,T,U,Z,?,Help): *press Enter*
Extend another (<Yes>/No)? *No*
Note: R50 and L10 can be used to go right 50, left 10, etc.




```
Extend By Distance Commands                                4/25/2006 16:41
#  - Number              Distance to draw or move
A# - Angle change        Rotates pointer by specified # of degrees (dd.mmss)
A  - Align               Rotates pointer to align with line or polyline segment
B  - Bearing/Azimuth/    Sets pointer direction by bearing/azimuth/angle in format:
     Angle               Qdd.mmss with Q-quadrant, d-degrees, m-minutes and s-seconds.
                         (Quadrant 1=NE, 2=SE, 3=SW, 4=NW, 5=AZ, 6=AL, 7=AR, 8=DL, 9=DR)
                         (e.g. 130.1005 is NE 30 deg, 10 min, 5 sec)
C  - Close               Close the polyline
D  - Draw mode           Actions draw or extend the line or polyline
E  - Extend to Edge      Extends to a boundary edge
I  - Input mode          Toggles distance input between decimal feet and feet-inches
L  - Left rotate         Rotates counter-clockwise 90 degrees
M  - Move mode           Actions only move the pointer
N  - Next                Moves pointer forward to next point
O  - Open                Open the polyline
P  - Previous            Moves pointer backward to previous point
R  - Right rotate        Rotates clockwise 90 degrees
S  - Switch direction Reverses the pointer direction
T# - Total distance      Sets current segement to specified distance
U  - Undo
Z  - Zoom mode           Toggles auto-zoom between on/off
?  - Info                Displays lengths of current polyline
Press <Enter>            Ends the routine
```

The result of using the Help (H) option

Pulldown Menu Location: Edit > Extend
Keyboard Command: extender
Prerequisite: An existing line or polyline with at least one segment from which to start.
File Name: \lsp\scadutil.arx

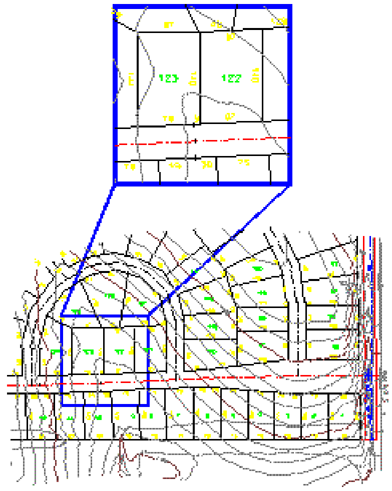
Break by Crossing Polyline

This tool is used to break drawing geometry at the edge of a polyline boundary. It provides options to change the layers of the interior and exterior geometry after it is broken.

First, select the boundary polyline. Only one can be selected. Then select the polylines and lines to be clipped. You will be prompted for options on specifying the layers for the newly broken geometry. Respond with a "Y" if you want to specify a new layer, then enter the new layer name. If the layer name does not exist, it will be created.

Prompts

Select the clip edge polyline: *pick a closed polyline*
Select the polylines and lines to be clipped.
Select Objects: *pick the entities to break*
Specify layer names for Inside segments (Yes/<No>)? *Yes*
Enter a layer name for the Inside segments <0>: *press Enter*
Specify layer names for Outside segments (Yes/<No>)? *Yes*
Enter a layer name for the Outside segments <0>: *Final*



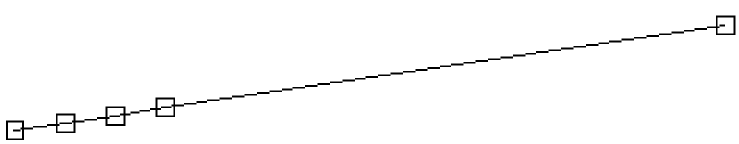
Pulldown Menu Location: Edit > Break
Keyboard Command: clipline
Prerequisite: A closed polyline
File Name: \lsp\poly3d.arx

Break Polyline at Specified Distances

This command allows you to pick a polyline and break it at a specified distances along the polyline. Following the prompts below, the beginning of the polyline in the illustration was broken into three 55-foot segments.

Prompts

Select polyline to break: *select polyline*
Total Distance: 779.429 This is the length of the polyline reported.
Distance Along Polyline For Break: *55.0*
Distance Along Polyline For Break (Enter to end): *110*
Distance Along Polyline For Break (Enter to end): *165*
Distance Along Polyline For Break (Enter to end): *press Enter*
3 polyline breaks created.



Pulldown Menu Location: Edit > Break
Keyboard Command: breakpl
Prerequisite: A polyline
File Name:

Break, Select Object, 2nd Point

>

This command allows you to break an object by selecting theobject, then the second break point. The first break point is the point whereyou select the object.

Prompts

1 Select object: **selectentity to break**

2 Specify second break point or[First point]: **select second break point**

Break, Select Object, Two Points

This command allows you to break an object by selecting theobject, then two points. First select the object, then the program will promptyou to select two points that define where the object will be broken.

Prompts

1 Select object: **selectentity to break**

2 Specify second break point or[First point]: **First**

3 Specify first break point: **pick first point**

4 Specify second break point: **pick second point**

Break at Intersection

This command will break a line, arc or polyline at the intersection of another line, arc or polyline. In many cases this command is used in conjunction with the *Area by Lines & Arcs* command. In order to get the correct area of a figure, it is often necessary to break it from adjoining lines.

Prompts

Select Line, Arc, or Polyline to Break
Select object: *select object to break*
[int on] Pick Intersection to break at: *pick intersection point*

Pulldown Menu Location: Edit > Break
Keyboard Command: breakat
File Name: \lsp\surv1.lsp

Break at Selected Point

This command allows you to break an object by selecting the object. Only one pick is necessary since TakeOff both selects the object and treats the selection point as the break point.

Prompts

1 Select object: select entity to break

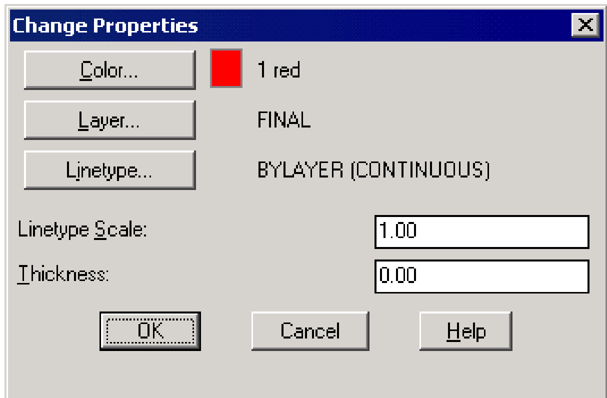
Select an object to break

Prerequisite: None

Keyboard Command: BREAK

Change Properties

This command allows you to change certain properties of existing objects.



1 In the Change Properties dialog box, you must choose the properties to modify.

- **Color**: This option allows you to change the color of the object.
- **Layer**: This option allows you to change the layer of the object.
- **Linetype**: This option allows you to change the linetype of the object.
- **Linetype Scale**: This option specifies the linetype scale factor for the new linetype.
- **Thickness**: This option specifies the distance to extrude the object above or below its elevation.

Note: The Properties command allows you to modify entity specific properties such as the radius of a circle or the height of a text entity.

Prerequisite: None

Keyboard Command: DDCHPROP

Rotate by Bearing

This command allows you to move objects about a base point by a given bearing.

Prompts

1 Select entities to rotate.

Select objects: **pick entities**

2 Base pivot point?

Pick point or point number: **pick a point**

3 Reference Bearing point?

Pick point or point number: **pick a point**

4 Azimuth/<New Bearing (Qdd.mmss)>: **enter a bearing**

Prerequisite: None

Keyboard Command: BROT

Standard Rotate

Function

This command allows you to move objects about a base point using a point as a rotation reference.

Prompts

1 Select objects: **pick entities**

2 Specify base point: pick point on screen as reference

3 Specify rotation angle or [Reference]: **rotate to desired location**

Prerequisite: None

Keyboard Command: ROTATE

Edit Text

This command allows you to edit text and attribute labels.

1 Select Text to Edit: **select the text**

You can modify text in provided text field.

Prerequisite: Text

Keyboard Command: EDITXT

Find and Replace Text

Function

With this command, you can find, replace, select, or zoom to text contained in the current drawing.

Keyboard Command: find
Prerequisite: Text

Text Enlarge/Reduce

This command will scale text entities up or down in size. The routine prompts for a scale multiplier and a selection set of text objects. If you want to enlarge the text enter a value greater than one. If you want to reduce text enter a decimal fraction such as .5. This would reduce the text size by 50%. This command is very useful if you have set up your drawing for one plotting scale and decide to change to a new plotting scale. The Change Text Size command can alternatively be used to set the text size to a specific value.

Pulldown Menu Location: Edit > Text
Prerequisite: Text entities to be changed
Keyboard Command: txtenl
File Name: \lsp\surv1.lsp

Text Explode To Polylines

This command converts the selected text into polylines. This function is generally used when preparing a plan view file for machine control, before using the *Write Polyline File* command.

Prompts

Select text to be EXPLODED.
Select objects: *select the text*
Substitute With Simple Font [<Yes>/No]? *Y*
1 text object(s) have been exploded to lines.
The line objects have been placed on layer 0.
Reading the selection set ...
Joining ...
Converting ...

Pulldown Menu Location: Edit > Text
Keyboard Command: textexp
Prerequisite: Text
File Names: \lsp\textexp.lsp, \lsp\poly3d.arx

Image Frame

This command controls whether TakeOff displays the image frame or hides it from view.

Because you select an image by clicking its frame, setting the image frame to off prevents you from selecting an image.

Prompts

1 Enter image frame setting [ON/OFF] <current>: **enter an option or Press Enter**

- **On**: Displays image frames so you can select images.
- **Off**: Hides image frames so you cannot select images.

Prerequisite: None

Keyboard Command: IMAGEFRAME

Image Clip

This command allows you to create new clipping boundaries for an image object.

Prompts

1 Select image to clip: select the edge of an image

2 Enter image clipping option [ON/OFF/Delete/New boundary] <New>: **enter an option or Press Enter**

The boundary you specify must be in a plane parallel to the image object.

- **On**: Turns on clipping and displays the image clipped to the previously defined boundary.
- **Off**: Turns off clipping and displays the entire image and frame. If you reclip the image while clipping is turned off, the program automatically turns clipping back on. The program prompts you to delete the old boundary even when clipping is turned off and the clipping boundary is not visible.
- **Delete**: Removes a predefined clipping boundary and redisplays the full original image.
- **New Boundary**: Specifies a new clipping boundary. The boundary can be rectangular or polygonal, and consists only of straight line segments. When defining a clipping boundary, specify vertices within the image boundary. Self-intersecting vertices are valid. Rectangular is the default option. If you use the pointing device to specify a point at the Enter Clipping Type prompt, the program interprets the point as the first corner of a rectangle.

3 Enter clipping type [Polygonal/Rectangular] <Rectangular>: **enter P or Press Enter**

- **Polygonal**: Uses specified points to define a polygonal boundary.

Specify first point: Specify a point

Specify next point or [Undo]: **specify a point or enter u**

Specify next point or [Undo]: **specify a point or enter u**

Specify next point or [Close/Undo]: **specify a point, or enter c or u**

You must specify at least three points to define a polygon.

If the image already has a clipping boundary defined, TakeOff displays the following prompt:

Delete old boundary? [No/Yes] <Yes>: **enter N or Press Enter**

If you choose Yes, the program redraws the entire image and the command continues; if you choose No, the command ends.

- **Rectangular**: Specifies a rectangular boundary by its opposite corners. TakeOff always draws the rectangle parallel to the edges of the image.

Specify first corner point: **specify a point**

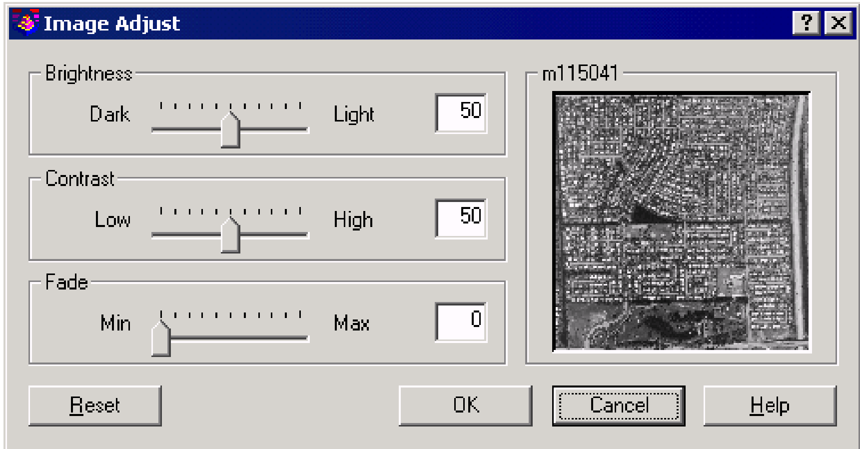
Specify opposite corner point: **specify a point**

Prerequisite: None

Keyboard Command: IMAGECLIP

Image Adjust

This command controls the display of the brightness, contrast, and fade values of images.



The Image Adjust dialog box controls how the image is displayed by adjusting the brightness, contrast, and fade settings of the selected image. Adjusting these values changes the display of the image but does not change the image file itself.

- **Brightness**: Controls the brightness, and indirectly the contrast, of the image. Values range from 0 through 100. The greater the value, the brighter the image and the more pixels that become white when you increase contrast. Moving the slider to the left decreases the value; moving the slider to the right increases the value.
- **Contrast**: Controls the contrast, and indirectly the fading effect, of the image. Values range from 0 through 100. The greater the value, the more each pixel is forced to its primary or secondary color. Moving the slider to the left decreases the value; moving the slider to the right increases the value.
- **Fade**: Controls the fading effect of the image. Values range from 0 through 100. The greater the value, the more the image blends with the current background color. A value of 100 blends the image completely into the background. Changing the screen background color causes the image to fade to the new color. In plotting, the background color for fade is white. Moving the slider to the left decreases the value; moving the slider to the right increases the value.
- **Image Preview**: Displays a preview of the selected image. The preview image updates dynamically to reflect changes to the brightness, contrast, and fade settings.
- **Reset**: Resets values for brightness, contrast, and fade to default settings (50, 50, and 0, respectively).

Prerequisite: None

Keyboard Command: IMAGEADJUST

Remove Groups

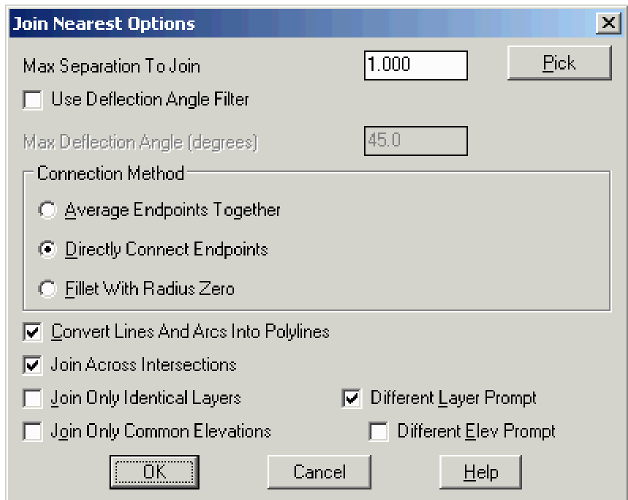
This command breaks up AutoCAD groups into more managable entities.

Prerequisite: Groups

Keyboard Command: rmgroup

Join Nearest

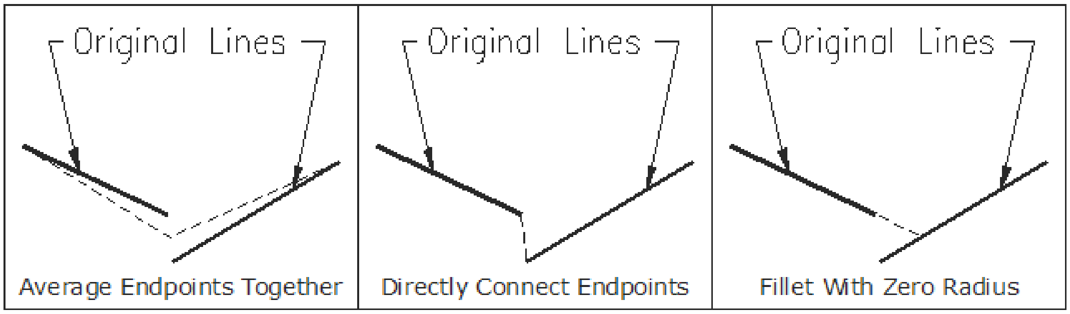
This command joins lines, arcs and/or polylines together. While AutoCAD's *PEDIT-Join* command requires the endpoints to match, Join Nearest will allow you to join entities whose endpoints do not exactly meet. You specify the maximum separation distance to join, along with other options, in the dialog box shown below. Also you can join many entities at once.



Max Separation to Join: Entities whose endpoints are spaced apart greater than this value will not be joined. You may use the pick button to specify this value by picking two points on the screen.
Max Deflection Angle (degrees): This option will not join any lines if the angle between them is greater than this angle in degrees.
Connection Method: Determines how to connect the endpoints. See the illustration below.

1. **Average Endpoints Together:** New vertex will be located at midpoint between two original endpoints (see illustration below on left).
2. **Directly Connect Endpoints:** Original endpoints are connected with new segment (see the middle illustration below).
3. **Fillet with Radius Zero:** Same as AutoCAD *FILLET* command using zero radius (see the illustration on right).

Convert Lines and Arcs Into Polylines: When checked, automatically converts lines and arcs into polylines. If not checked, lines and arcs are joined but remain separate entities.
Join Across Intersections: This option applies to cases where more than two linework endpoints come together such as a Y intersection. In these cases, there are multiple possible connections. When this option is on, the program will automatically choose one of the possible connections. Otherwise, the program will not connect any of them.
Join Only Identical Layers: When checked, only entities on the same layer will be joined.
Join Only Common Elevations: When checked, only endpoints located on the same elevation will be joined.
Different Layer Prompt: When Join Only Identical Layers is off, then this option will prompt for which layer to use when it finds a connection between two different layer names.
Different Elevation Prompt: When Join Only Common Elevations is off, then this option will prompt for which elevation to use when it finds a connection between two different elevations.



Pullown Menu Location: Edit
Keyboard Command: nearjoin
Prerequisite: Lines or polylines to be joined
File Names: \lsp\nearjoin.lsp, \lsp\poly3d.arx

3D Entity to 2D

This command changes a 3D Line, Arc, Circle, Polyline, Insert or Point to 2D, i.e. an entity with the elevations of the endpoints at the same Z coordinate. When the program detects a 3D polyline with all vertices with the same elevation, there is an option to convert to a 2D polyline with this elevation. Otherwise, the entered elevation here is used.

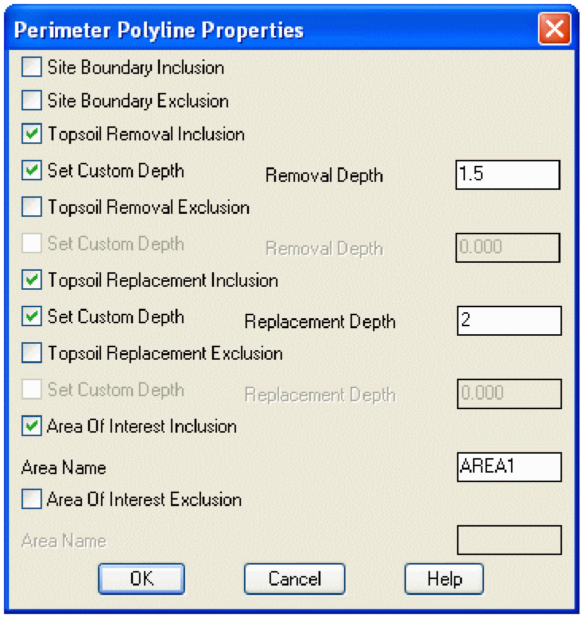
Prompts

Select/<Enter Elevation <0.00>: *press Enter*
Select Lines, Arcs, Circles, Polylines, Inserts and Points for elevation change.
Select objects: *pick a 3D polyline*
3DPOLY to 2DPOLYLINE
Number of entities changed> 1

Pulldown Menu Location: Edit
Keyboard Command: 3dto2d
Prerequisite: None
File Name: \lsp\3dto2d.lsp

Perimeter Polylines Properties

This command allows you to control the properties of any perimeter polyline (Note: Perimeter polylines also have to be closed polylines). Select a polyline and the following dialog appears. Here you can define the functionality of the polyline in regards to, the Site Boundary, Areas of Interest, and Topsoil Removal/Replacement. These properties can also be set separately using the Boundary Polyline, Areas of Interest, and Topsoil Removal/Replacement commands found under the Tools menu of Carlson Takeoff.



Prerequisite: a polyline
Keyboard Command: perim_prop

Entities to Polylines

This command converts selected lines, arcs, circles, 3DFaces, ellipses, splines, multilines, regions and solids into individual polylines. Use *Join Nearest* to convert adjoining lines and arcs into continuous polylines.

Prompts

Select lines, arcs, circles, 3DFaces, ellipses, splines, multilines, regions and solids to convert.
Select objects: *select entities*

Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: topline
Prerequisite: lines, arcs or other entities to convert
File Name: \lsp\poly3d.arx

Reverse Polyline

This command reverses the order of the line and/or arc segments of a *POLYLINE*. This can be useful in conjunction with the commands *Station Polyline*, *MXS by Polyline*, *Profile from Surface Model* or *CL File from Polyline*, since the polyline must be plotted in the direction of increasing stations. If it is more convenient to draft a polyline in one direction do so and then use the *Reverse Polyline* command to change it's order. Temporary arrows along the polyline are drawn to graphically show the new polyline direction.

Prompts

Select the Polyline to Reverse: *pick a point on polyline*

Puldown Menu Location: Edit > Polyline Utilities
Keyboard Command: revpline
Prerequisite: A polyline
File Names: \lsp\revpline.lsp, \lsp\poly3d.arx

Reduce Polyline Vertices

This command removes points from a polyline, without significantly changing the polyline. The offset cutoff is the maximum amount that the polyline can move horizontally and vertically when removing a point. For example, in a polyline with three points in a straight line, the middle point can be removed without changing the polyline. This command is explained further in the *Triangulate & Contour* command.

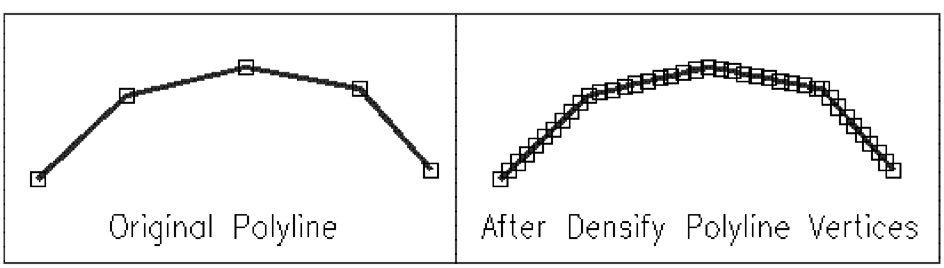
Prompts

Enter the offset cutoff <0.1>: *.5*
Select polylines to reduce.
Select objects: *pick polylines*
Processed polylines: 1
Total number of vertices: 10
Number of vertices removed: 1

Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: reduce
Prerequisite: A polyline
File Name: \lsp\tri4.arx

Densify Polyline Vertices

This command adds vertices to the selected polylines at the specified interval. These points are interpolated between existing points in the polyline. This command is the opposite of Reduce Polyline Vertices.



Prompts

Select polylines to densify.
Select objects: *select polylines*

Point interval <10.0>: *press Enter*
Testing Entity> 1
Added 17 points to 1 polyline.

Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: densepl
Prerequisite: A polyline
File Name: \lsp\poly3d.arx

Smooth Polyline

This command smooths the selected polylines using a modified Bezier method that makes the smooth polyline pass through all the original points and only smooths between the original points. The looping factor controls smoothing amount. A higher factor gives more looping. This command is explained further in the Surface menu section.

Prompts

Enter the looping factor (1-10) <5>: *7*
Enter the offset cutoff <0.05>: *press Enter* This is the same reducing filter described above.
Select polylines to smooth.
Select objects: *pick polylines*
Smoothed 1 PolyLines
Total original vertices: 9 Total final vertices: 50

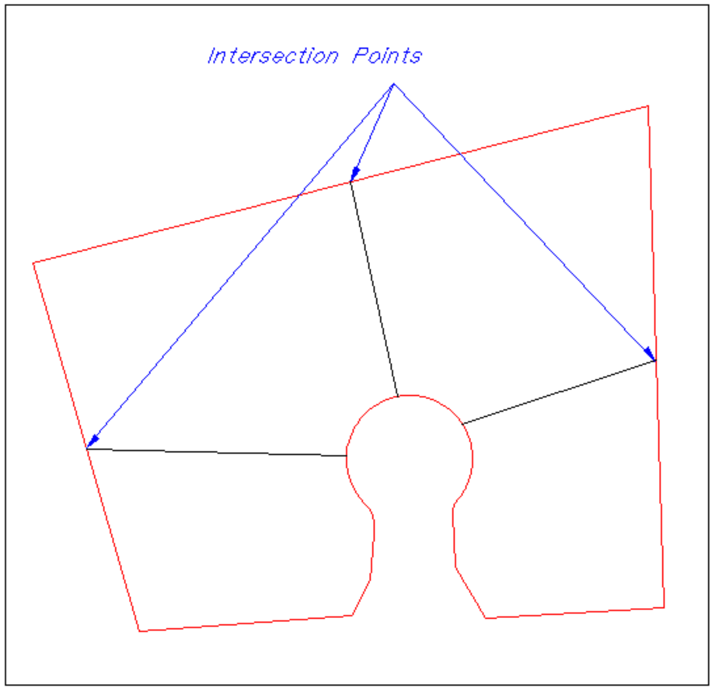
Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: smoothpl
Prerequisite: A polyline
File Name: \lsp\tri4.arx

Add Intersection Points

This command adds points into lines or polylines where there are intersections. This can be useful for other commands such as Auto-Annotate. For example in the drawing shown, Add Intersection Points adds points to the boundary polyline where the lot lines intersect. Then Auto Annotate for the boundary polyline will label the boundary distance along each lot. This routine does not add intersection points on arcs.

Prompts

Select lines and polylines to check.
Select objects: *pick lines or polylines*
Reading the selection set ...
Adding intersection points ...
Added 3 intersection points.



Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline
Keyboard Command: addint
Prerequisite: Polylines or lines
File Name: \lsp\poly3d.arx

Add Polyline Vertex

This command adds points into a polyline. First you select the polyline to modify. The existing polyline vertices are marked and then you can pick or enter the coordinates for the new point(s). A new point is inserted into the polyline at the nearest polyline segment. On a 3D polyline, the elevation of the new vertex will be calculated for you. You can continue to pick points to add. Press Enter when you are done.

Prompts

Select polyline to add to: *pick a polyline*
Pick or enter point to add: *pick a point*
Select polyline to add to: *press Enter to end*

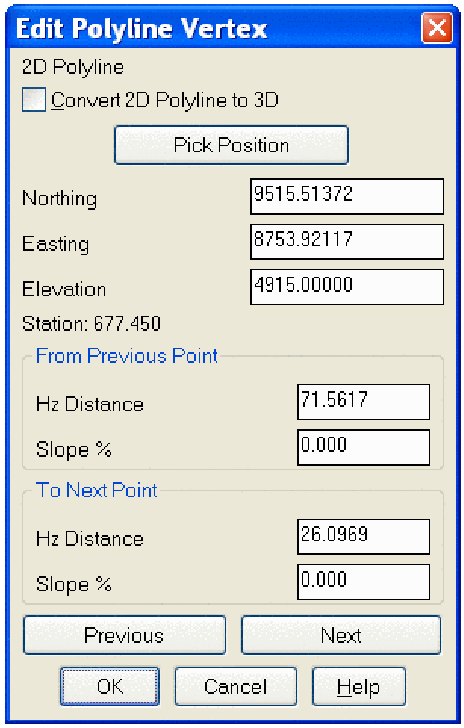
Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline
Keyboard Command: addpl
Prerequisite: A polyline
File Name: poly3d.arx

Edit Polyline Vertex

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution, you will be asked to select a polyline to edit. Upon selection, a temporary marker will be placed at all of the vertices of

the polyline, making them easy to distinguish. You must then pick near the vertex you wish to edit. The following dialog appears.

At the top of the dialog it identifies the type of polyline as being 2D or 3D. In the case of 2D polylines, it allows you convert the polyline. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example, if you change the horizontal distances, the coordinates will change.



Prompts

Select polyline to edit: *pick a polyline*
Pick point on polyline to edit: *pick a point to be modified*
Edit Polyline Vertex dialog *click "Pick Position"*
Pick vertex position: *pick a new location for the vertex*
Edit Polyline Vertex dialog *click OK*
Make changes as needed. You will see the polyline vertices relocated based upon the new picked positions and coordinate changes. Use Previous and Next to move along the polyline. Note the dialog values changing.
Select polyline to edit (Enter to end): *press Enter to end*

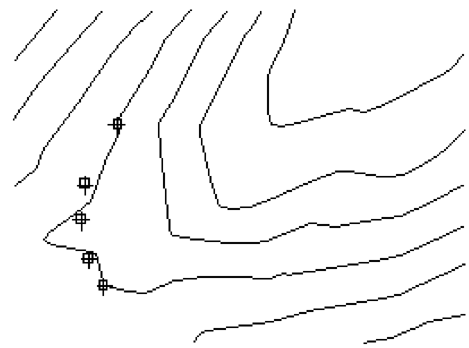
Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: editpl
Prerequisite: A polyline
File Name: \lsp\poly3d.arx

Edit Polyline Section

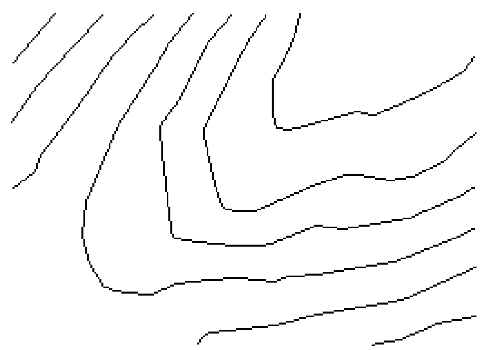
This command revises a segment of a polyline. Begin by picking a point on the polyline where you want to start editing. Then pick new points for the polyline. When finished picking new points press Enter, and then pick a point on the polyline to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

Prompts

Select polyline to edit: *pick the polyline at the place to start editing*
Pick intermediate point (Enter to End): *pick a point*
Pick intermediate point ('U' to Undo, Enter to End): *pick a point*
Pick intermediate point ('U' to Undo, Enter to End): *press Enter*
Pick reconnection point on polyline: *pick the polyline at the place to join*



Edit this contour by picking new points



Contour with segment replaced with new points

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline
Keyboard Command: editpl2
Prerequisite: Polylines
File Name: \lsp\poly3d.arx

Change Polyline Width

This command sets the width of the selected polylines. In later versions of AutoCAD, the command *PEDIT* can also modify the width of multiple polylines.

Prompts

New width <1.0>: *2*
Select Polylines/Contours to change width of:
Select objects: *pick polylines*

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline
Keyboard Command: cwidth
Prerequisite: A polyline
File Name: \lsp\surv1.lsp

Set Polyline Origin

This command sets the starting vertex of a closed polyline. Simply pick the polyline and then pick near the point to set as the starting point.

Prompts

Select Polyline: *pick a polyline*
Pick Near New Origin Point: *pick a point on the polyline to be the starting point*
Processing ...
Select Polyline: *press Enter*

Pulldown Menu Location: Edit > Polyline Utilities > Edit Polyline
Keyboard Command: plchgorg
Prerequisite: A closed polyline
File Name: \lsp\plchgorg.lsp

Close Polyline

This command allows you to close a selection set of open polylines.

Prerequisite: Open polyline(s).

Keyboard Command: CLOSEPL

Open Polyline

This command allows you to open a selection set of closed polylines.

Prerequisite: Closed polyline(s).

Keyboard Command: OPENPL

Remove Polyline Arcs

This command replaces arc segments in polylines with chords. Removing arcs is a prerequisite to some Carlson commands that don't handle arcs, such as *Break by Closed Polyline* and *Make 3D Grid File*. This process can add many vertices to the polyline. The Offset cutoff is the maximum any point on the arc will be allowed to shift.

Prompts

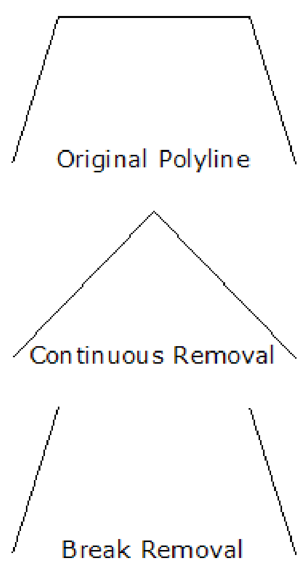
Select polylines to remove arcs from.
Select objects: *pick polylines*

Offset cutoff <0.5>: *press Enter*

Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline
Keyboard Command: rmarc
Prerequisite: polyline with arcs
File Name: \lsp\poly3d.arx

Remove Polyline Segment

This command removes the user specified segment from a polyline. A polyline segment is the section between two vertices of the polyline. There are two options for removing the segment. Either the two vertices of the removed segments are averaged together to keep polyline continuous, or the segment is left missing in the polyline, which creates two separate polylines. The keywords Continuous and Break respectively identify these two options. The first image is of the Original Polyline. The second is with the Continuous Removal option. The third is using the Break Removal option.



Prompts

Break polyline at removal or keep continuous (Break/<Continuous>)? *press Enter*
Select polyline segment to remove: *pick point on polyline*
Select polyline segment to remove: *press Enter to end*

Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline
Keyboard Command: removepl
Prerequisite: A polyline
File Names: \lsp\removepl.lsp, \lsp\poly3d.arx

Remove Polyline Vertex

This command removes vertices from a polyline. First you select the polyline to modify. The existing polyline vertices are marked and then you pick near the vertex you wish to delete. You can continue to pick vertices to delete, press Enter when you are done.

Prompts

Select polyline to remove from: *pick point on polyline*
Pick point to remove: *pick point*
Pick point to remove (Enter to end): *press Enter to end*

Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline
Keyboard Command: rmvertex
Prerequisite: A polyline
File Name: \lsp\poly3d.arx

Tag Hard Breakline Polylines

Function

This command tags polylines with a description so that Triangulate & Contour can identify these polylines as hard breaklines. The tag is invisible and doesn't change the polyline. Triangulate & Contour will not smooth the contours as they crosses these hard breaklines. For example you could tag 3D polylines that represent a wall so that the contours go straight across the wall without smoothing curves.

Prompts

Select hard breaklines.

Select objects: *select polylines*

Prerequisite: Polylines.

Keyboard Command: hardbrk

Untag Hard Breakline Polylines

Function

This command removes description tags from polylines. These tags are used by Triangulate & Contour to identify polylines as hard breaklines. Contours are not smoothed as they cross these hard breaklines. This routine untags polylines so that contours are smoothed across them.

Prompts

Select polylines to remove hard breakline tag from.

Select objects: *select polylines*

Prerequisite: Polylines with hard breakline tag.

Keyboard Command: softbrk

Remove Duplicate Polylines

This command analyzes the selected polylines and erases any duplicate polylines found. They must be exactly the same for one to be deleted.

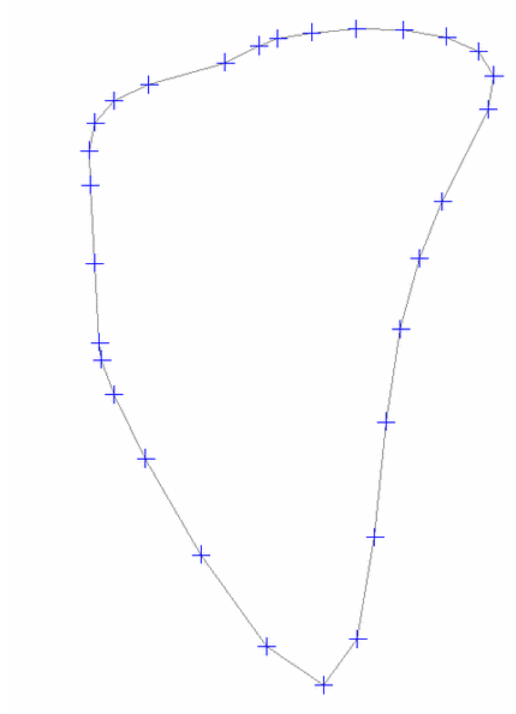
Prompts

Select lines, arcs and polylines to process.
Select objects: *select linework to process*
Reading the selection set ...
Removed 1 duplicate linework entities.

Pulldown Menu Location: Edit > Polyline Utilities > Remove Polyline
Keyboard Command: rmduplwork
Prerequisite: Polylines that have duplicates
File Name: poly3d.arx

Draw Polyline Blips

This command will draw temporary markers, "blips", at each polyline vertex. This allows you to identify the actual location of each vertex. The Blips are temporary. Any change to the viewport (pan, zoom, regen) will make the blips disappear. In later versions of AutoCAD, you can also click on the polyline to activate the grips which will remain visible during and after viewport changes.



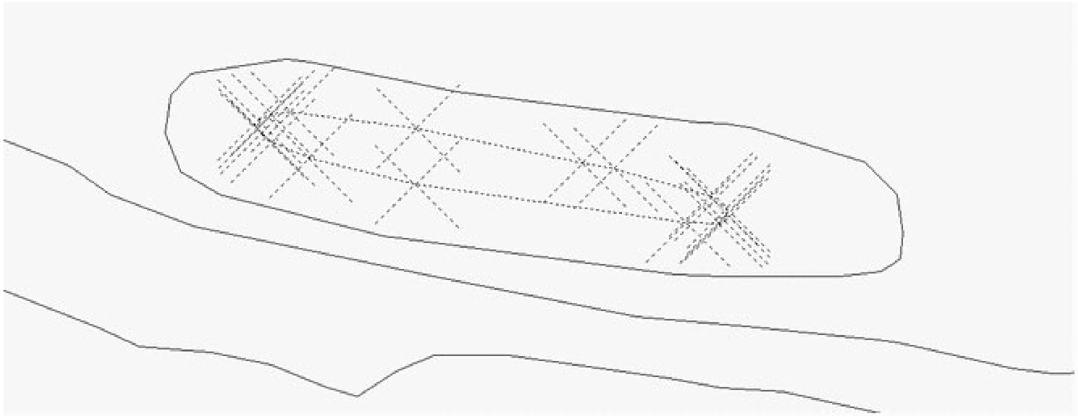
Prompts

Select polylines to draw blips.
Select objects: *select polyline(s)*

Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: plblip
Prerequisite: A polyline
File Name: \lsp\poly3d.arx

Check Elevation Range

This command analyzes a selection set of polylines, and highlights the ones that fall outside of a specified elevation range. There is an option to set the polylines that are outside of the range to zero. Every polyline vertex that is outside of the range will be highlighted with an X.



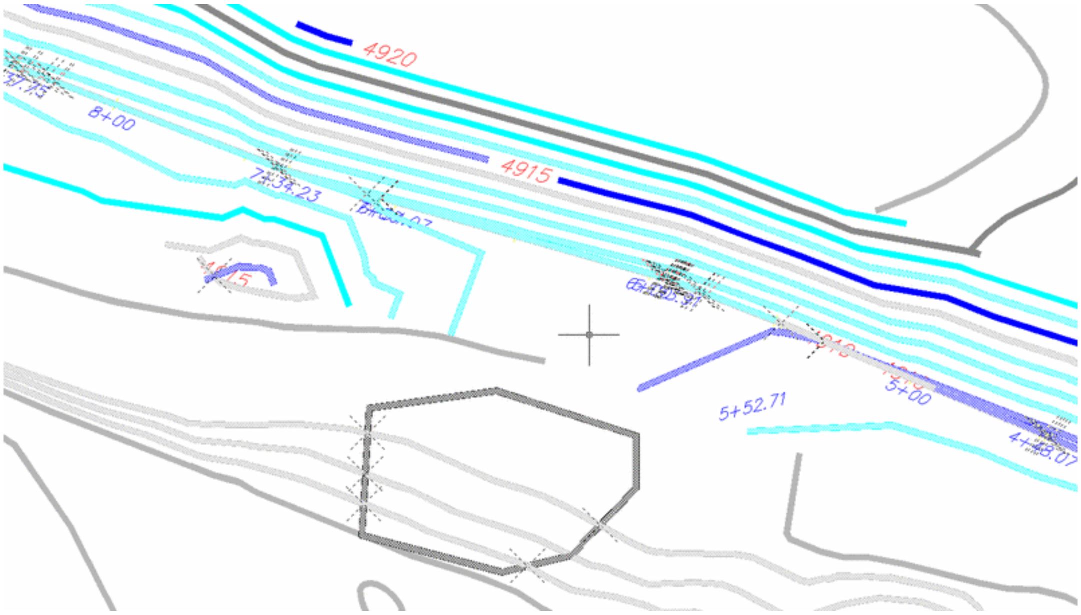
Prompts

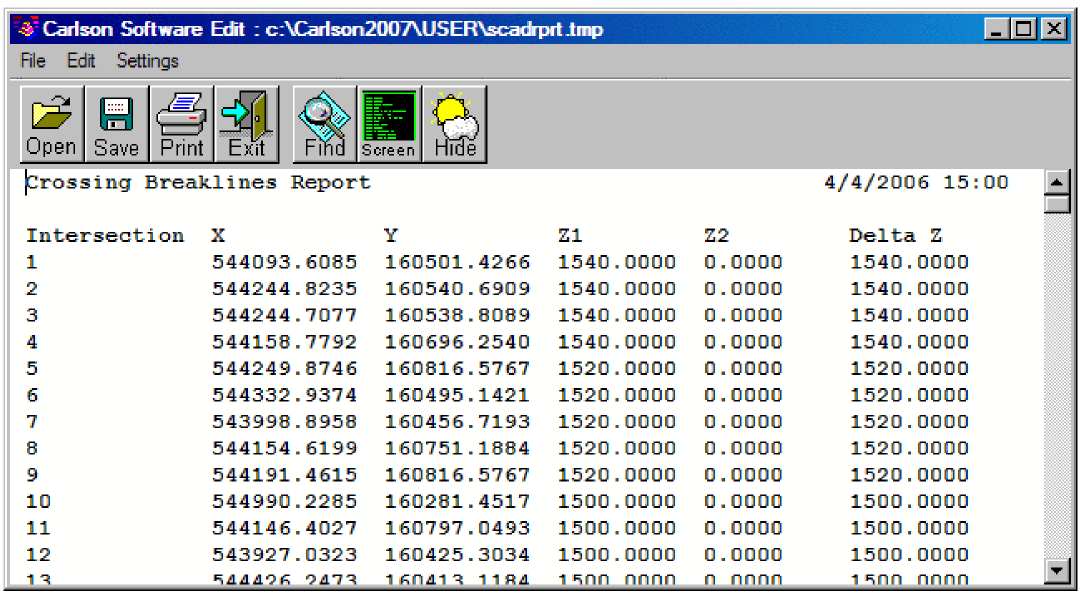
Enter elevation range minimum: *0*
Enter elevation range maximum: *4900*
Select polylines to check.
Select objects: *pick polylines to process*
Found 1 polylines outside of elevation range.
Set polylines outside elevation range to zero elevation [Yes/<No>]? *N*

Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: checkpl
Prerequisite: Polylines with elevations
File Names:

Highlight Crossing Plines

This command highlights selected polylines that are crossing in the drawing and have different elevations at the crossing. Every intersection point where the polylines cross are marked with a temporary X. A report is provided at the end where the X and Y of the intersection points are displayed with the two Z values and the Z difference. The command has the ability to repair crossing polylines by inserting a vertex in each polyline at the intersection and assigning a common elevation at this intersection.





Crossing Breaklines Report 4/4/2006 15:00

Intersection	X	Y	Z1	Z2	Delta Z
1	544093.6085	160501.4266	1540.0000	0.0000	1540.0000
2	544244.8235	160540.6909	1540.0000	0.0000	1540.0000
3	544244.7077	160538.8089	1540.0000	0.0000	1540.0000
4	544158.7792	160696.2540	1540.0000	0.0000	1540.0000
5	544249.8746	160816.5767	1520.0000	0.0000	1520.0000
6	544332.9374	160495.1421	1520.0000	0.0000	1520.0000
7	543998.8958	160456.7193	1520.0000	0.0000	1520.0000
8	544154.6199	160751.1884	1520.0000	0.0000	1520.0000
9	544191.4615	160816.5767	1520.0000	0.0000	1520.0000
10	544990.2285	160281.4517	1500.0000	0.0000	1500.0000
11	544146.4027	160797.0493	1500.0000	0.0000	1500.0000
12	543927.0323	160425.3034	1500.0000	0.0000	1500.0000
13	544426.2473	160413.1184	1500.0000	0.0000	1500.0000

Prompts

Select polylines to check.
Select objects: *pick polylines to process*
Ignore zero elevations [<Yes>/No]? *press Enter for Yes to filter out polylines at zero elevation*
Reading points ... 1677
Finding points on breaklines ...
19 crossing polylines are highlighted.
Use Report Formatter [Yes/<No>]? *press Enter for No.* Use the Report Formatter to customize the report layout or export to Excel.

Minimum delta Z to report <0.0>: *2*
Add polyline vertices at intersections [Yes/<No>]? *Y*
Set 3D polyline to crossing contour elev or average elevs [Set/<Average>]? *press Enter for Average.* The Set option applies to crossing polylines where one polyline is a 3D polyline with varying elevations and the other polyline is a contour polyline with a fixed elevation. For this case, the Set method will hold the elevation of the contour polyline and set the 3D polyline elevation to match the contour. The Average method sets the elevation of the intersection point as the average of the crossing polyline elevations at that point.
Maximum delta Z to average <1.0>: *press Enter.* This option will only add the intersection point with the averaged elevation if the elevation difference is less than this tolerance.

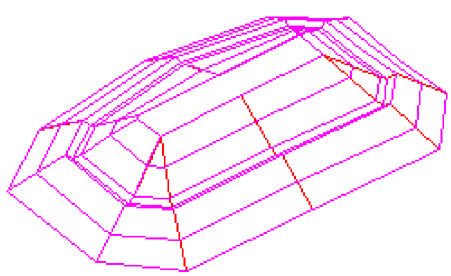
Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: xing_plines
Prerequisite: Polylines with elevations

Offset 3D Polyline

This command allows you to offset a 3D polyline entity in both the horizontal and vertical directions. There are three offset methods. The Interval method applies one horizontal and one vertical offset to all the vertices of the polyline. The Constant method has a horizontal offset and sets the elevation of the polyline to one constant elevation. The Variable method allows you to specify each horizontal and vertical offset individually either by polyline segment or for each point. The vertical offset can be specified by actual vertical distance, percent slope or slope ratio.

Prompts

Enter the offset method [<Interval>/Constant/Variable]: *press Enter*
Vertical/<Horizontal offset amount>: *15*
Percent/Ratio/Vertical offset amount <0>: *10*
Select a polyline to offset (Enter for none): *select a 3D poly*
Select side to offset: *pick a point*
Select a point on the graphics screen that is in the direction of the side of line to offset.
Select a polyline to offset (Enter for none): *press Enter*



Pulldown Menu Location: Edit > 3D Polyline Utilities
Keyboard Command: offset3d
Prerequisite: Plot the *3DPoly* lines to use for selection.
File Name: \lsp\poly3d.arx

Fillet 3D Polyline

This command fillets two segments of a 3D polyline with the given radius. AutoCAD's *FILLET* command does not support 3D Polyline entities. Since 3D polylines cannot have arcs, this command draws the fillet arc as a series of short chords. The elevations along the curve are interpolated from the 3D polyline.

Prompts

Fillet corner of a polyline or intersection of two polylines [<Corner>/Intersection]? *press Enter*
Enter fillet radius <10.00>: *press Enter*
Select a corner point on polyline: *pick 3D polyline near meeting point of two segments*
Select a corner point on polyline: *pick 3D polyline near meeting point of two segments*
Select a corner point on polyline: *press Enter* (to end command)

Pulldown Menu Location: Edit > 3D Polyline Utilities
Keyboard Command: fillet3d
Prerequisite: 3D polyline
File Name: \lsp\poly3d.arx

Join 3D Polyline

This command joins *3DPOLY* entities into a single 3D polyline entity.

Prompts

Select the 3D polyline to join: *pick a 3D polyline*
Select the other 3D polyline to join: *pick a 3D polyline that has a common endpoint with the first*
3 segments added to the polyline.

Pulldown Menu Location: Edit > 3D Polyline Utilities
Keyboard Command: join3d
Prerequisite: Plot the *3DPoly* lines to use for selection
File Names: \lsp\join3d.lsp, \lsp\poly3d.arx

Break 3D Polyline by Surface

Function

This command breaks 3D polylines at the intersection with a surface.

Prompts

Select Surface
Select polylines to clip.
Select objects: *pick the 3D polylines*
Erase polyline below surface [<Yes>/No]? *press Enter* If you answer yes, the segments of the polylines below the surface will be erased from the intersection, if any, of the polyline with the surface. Otherwise the polylines will

only be broken into separate polylines at the intersection.

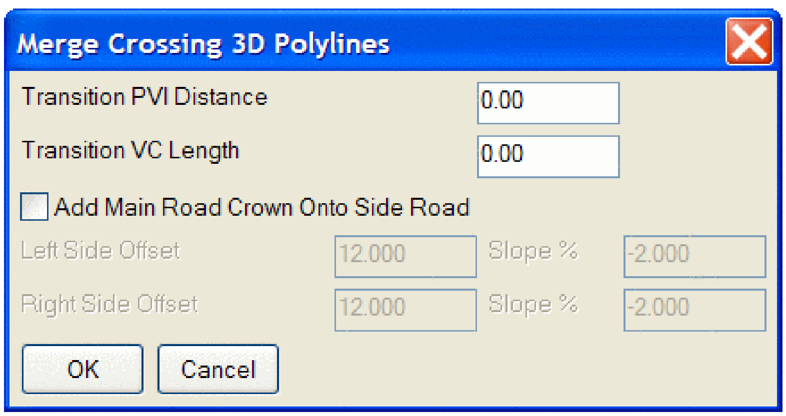
After Break 3D Polyline by Surface

Before Break 3D Polyline by Surface

Keyboard Command: surfbreak
Prerequisite: 3D Polylines to break and a surface.

Merge Crossing 3D Polylines

This command works with 2 crossing 3D polylines, adding one or more vertices to one of them at the virtual point of intersection to match the elevation of the other. The 3D polyline that is vertically unchanged is referred to as the "Main 3D polyline", the 3D polyline that is edited is referred to as the "Side 3D polyline." The command uses the 2 vertices on the Main 3D polyline on either side of the virtual intersection to determine an interpolated elevation on the Main 3D polyline at the point of virtual intersection, and adds a vertex on the Main 3D polyline at that location with the calculated elevation, but the vertical characteristics of the Main 3D polyline are otherwise unchanged. The Side 3D polyline gets a new vertex at the virtual intersection with the same interpolated elevation, thereby changing it's vertical definition as much as necessary to match. The characteristics of the transition are controlled by the settings in the Merge Crossing 3D Polylines dialog box.



Transition PVI Distance: This option creates 2 additional vertices on the Side 3D polyline, each at the specified distance from the virtual intersection, and both with the same elevation as the vertex at the virtual intersection, essentially creating a flat section.

Transition VC length: This option creates a vertical curve for the transition, passing through the interpolated elevation at the virtual intersection. The start of the vertical curve is the specified value from the virtual intersection, as is the end, so the overall length of the entire vertical curve is actually twice the value specified in the dialog box.

Add Main Road Crown Onto Side Road: This option creates the transition by assuming the Main 3D

polyline is a crowned roadway, and creates corresponding additional vertices on the Side 3D polyline.

Prompts

Select the Main 3D polyline: *pick the 3D polyline that will determine the crossing elevation, but will remain essentially unchanged*
Select the Side 3D polyline: *pick the 3D polyline that is be changed to match the Main 3D polyline elevation at the virtual intersection*

Adjust variables as desired in Merge Crossing 3D Polylines dialog box, pick OK.

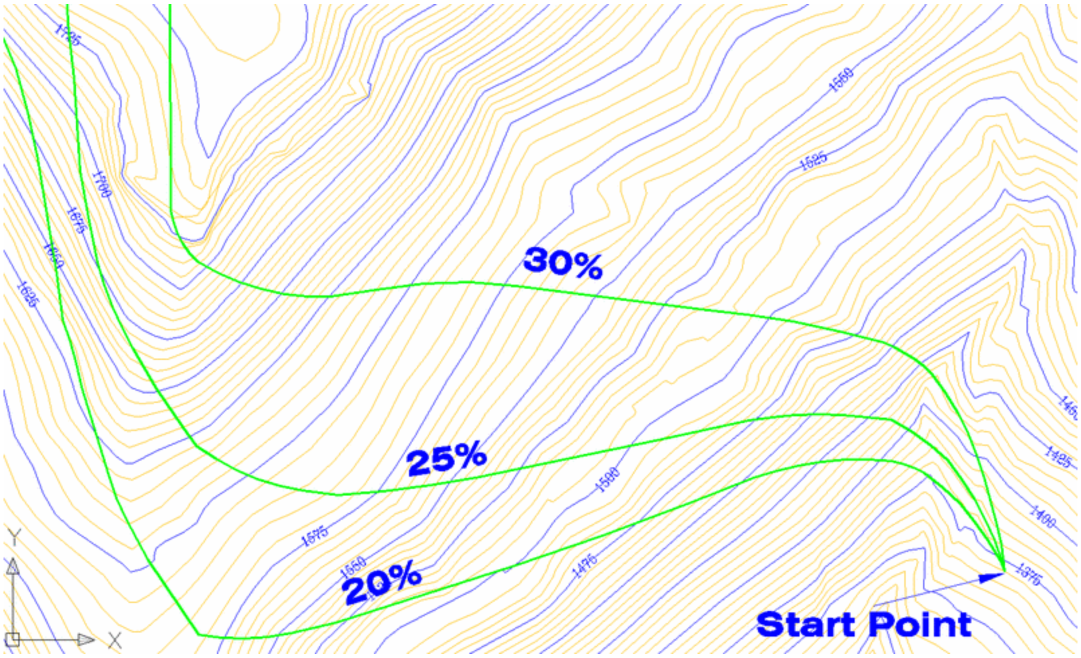
Pulldown Menu Location: 3D Data
Keyboard Command: merge3d
Prerequisite: 2 crossing 3D polylines
File Name: \lsp\eworks.arx

3D Polyline by Slope on Surface

This command creates a 3D polyline at a user-specified slope. The user picks the starting point and then the polyline continues along the surface at the slope until it reaches a point where the maximum slope at the point is less than the design slope. The surface is defined by a grid or TIN file which must be created before running this routine. Applications for this command include designing haul roads or ditches.

Prompts

Enter the polyline layer <SLOPE_ROAD>: *press Enter*
Select the Grid File dialog
Reading row> 51
Extrapolate grid to full grid size (Yes/<No>)? *Y*
Limiting length for polyline (Enter for none):
Pick origin point of 3D polyline: *pick a starting point*
Direction of 3D Polyline (<Up>/Down)? *press Enter*The slope must go either uphill or downhill.
Direction of 3D Polyline facing up slope (<Left>/Right)? *R* Imagine facing uphill. Do you want the polyline to go to the left or right?
Enter the design slope: *10* This value is in percent slope.



Pulldown Menu Location: 3D Data
Keyboard Command: surfpl
Prerequisite: Existing surface file
File Names: \lsp\slope_rd.lsp, \lsp\cntr_grd.arx

Add Points At Elevation

This command inserts vertices into a 3D Polyline at a specific elevation, or elevation interval, by interpolating between existing elevations in the polyline.

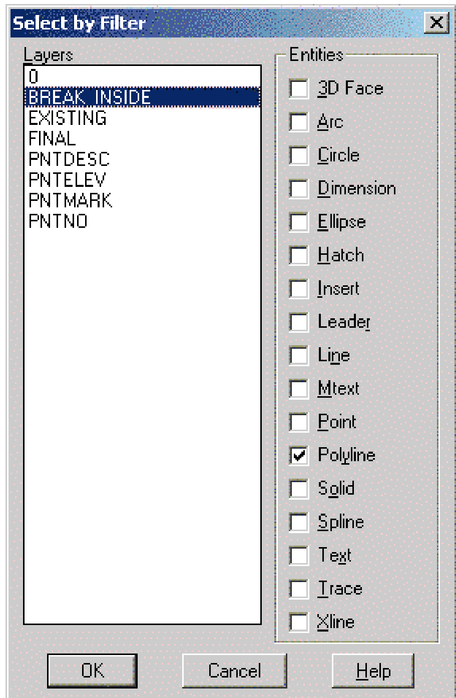
Prompts

Add single elevation or elevation interval [Single/<Interval>]? *press Enter*
Enter Elevation Interval: 50
Select 3D polylines to process. *pick 3D polyline(s)*
Select objects: 1 found
Select objects:
Processing polylines ...
Added 10 points to polylines.

Pulldown Menu Location: Edit > 3D Polyline Utilities
Keyboard Command: addplz
Prerequisite: 3D Polylines
File Name: \lsp\poly3d.arx

Select by Filter

This command can be used to build a selection set of objects inside a drawing based on layer and entity type. When the tool is executed it displays the following dialog. Select the layer(s) on the left you wish to select, then turn on the toggle(s) for the entity types to consider. The tool then builds a selection set of those objects that resides on those layers. When you execute your command following this selection building process, when you are prompted to select objects simply enter "P" for previous.



Pulldown Menu Location: Edit > Selection Sets
Keyboard Command: fsel
Prerequisite: None
File Name: \lsp\fsel.fas

Select by Elevation

This command builds a selection set of entities that are greater than, less than or in between a specified elevation that you enter in on the command line. Entities selected, based upon this elevation criteria, go into a selection set. With the Window selection method, the entities must be entirely inside of the inclusion area to be included in the selection set. With the Crossing selection method, an entity is added to the selection set if any part of the entity is inside the inclusion area.

Prompts

Select by greater, less or between elevations [<Greater>/Less/Between]? *press Enter*
Enter elevation for greater than: *19*
Ignore zero elevations [<Yes>/No]? *press Enter*
Select objects to build selection set. *pick objects*

Processing selection set ...
Built selection of 120 objects for elev more than 19.00.
To use type 'P' at Select objects: prompt.

Pulldown Menu Location: Edit > Selection Sets
Keyboard Command: zselect
Prerequisite: Entities
File Name: \lsp\volcalc.arx

Select by Area

This command builds a selection set using inclusion and/or exclusion closed polylines. Entities within the inclusion polylines are selected and entities within the exclusion polylines are not selected. With the Window selection method, the entity must be entirely inside the inclusion area and entirely outside the exclusion area to be included in the selection set. With the Crossing selection method, an entity is added to the selection set if any part of the entity is inside the inclusion area.

Prompts

Select the Inclusion perimeter polylines or ENTER for none:
Select objects: *pick the closed polyline*
Select objects: *press Enter*
Select the Exclusion perimeter polylines or ENTER for none.
Select objects: *press Enter*
Type of selection (Window/<Crossing>)? *press Enter*
Select objects to build selection set.
Select objects: *All* These selected objects are checked with the inclusion/exclusion polylines.
Select objects: *press Enter*
Built selection set with 43 objects.
Command: *Erase*
Select objects: *P* To use previous selection set created by Select by Area.
43 found
Select objects: *press Enter*

Pulldown Menu Location: Edit > Selection Sets
Keyboard Command: ssgetarea
Prerequisite: Closed perimeter polylines
File Name: \lsp\volcalc.arx

View Menu 4

Redraw

This command refreshes the display in the current viewport.

Prerequisite: None

Keyboard Command: R

Regen

This command regenerates the drawing and refreshes the current viewport.

Prerequisite: None

Keyboard Command: REGEN

Zoom - Window

This commands zooms to display an area you specify by two opposite corners of a rectangular window.

Prerequisite: None

Keyboard Command: ZOOM, W

Zoom - Dynamic

This command zooms to display the generated portion of the drawing using a view box. The view box represents your viewport, which you can shrink or enlarge and move around the drawing. Positioning and sizing the view box pans or zooms the viewport, filling it with the image inside the view box.

Prerequisite: None

Keyboard Command: ZOOM, D

Zoom - Previous

This command zooms to display a previous view. You can restore up to 10 previous views.

Prerequisite: None

Keyboard Command: ZOOM, P

Zoom - Center

This command zooms to display a window you define by picking a center point and a magnification value or height. A smaller value for the height increases the magnification. A larger value decreases the magnification.

Prompts

1 Specify center point: **pick a point**

2 Enter magnification or height <226.66>: **enter a value**

Prerequisite: None

Keyboard Command: ZOOM, C

Zoom - Extents

This command zooms to display the drawing extents. You can use Zoom Extents transparently, but it always regenerates the drawing.

Prerequisite: None

Keyboard Command: ZOOM, E

Zoom In

This command increases the zoom factor of the current viewport by a factor of 2.0.

Prerequisite: None

Keyboard Command: ZOOM, 2.0x

Zoom Out

This command decreases the zoom factor of the current viewport by a factor of 0.5.

Prerequisite: None

Keyboard Command: ZOOM, 0.5x

Zoom Points

This command centers the screen to a user-specified point. The point can be specified by either the point number or description. The command searches the current coordinate (.CRD) file. Besides centering the screen, the magnification can also be changed. The default value is the current magnification. To zoom in, enter a smaller value and to zoom out, enter a greater value.

Prompts

Find by point number or description [<Number>/Desc]? *N*
Point number or range of point numbers to find <1>: 2079
We want to find point number 2079
Magnification or Height <179.50>: *press Enter*

Accept the default zoom magnification

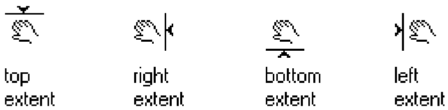
Pulldown Menu Location: View
Keyboard Command: zoompnt
Prerequisite: A .CRD file
File Names: \lsp\fpoint.lsp, \lsp\crdutil.arx

Pan

This command moves the drawing display in the current viewport. The cursor changes to a hand cursor. By holding down the pick button on the pointing device, you lock the cursor to its current location relative to the viewport coordinate system. The drawing display is moved in the same direction as the cursor.

hand cursor

When you reach a logical extent (the edge of the drawing space), a bar is displayed on the hand cursor on the side where the extent has been reached. Depending on whether the logical extent is at the top, bottom, or side of the drawing, the bar is either horizontal (top or bottom) or vertical (left or right side).



When you release the pick button, panning stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again to pan the display from that location.

To stop panning at any time, press Enter or ESC.

Prerequisite: None

Keyboard Command: P

Twist Screen Standard

This command allows you to "twist" the screen's orientation so that a direction other than North is toward the top of the screen and the drawing. It does not do a coordinate rotation, and it leaves the database unchanged. The ROTATE and MOVE commands in the Edit menu can be used to do a coordinate rotation and translation.

This command prompts you for the twist angle, then adjusts the screen and crosshairs to that angle. The twist angle is always measured counterclockwise, with 0 degrees at the east/right.

Prerequisite: None

Keyboard Command: TWIST1

Twist Screen Line

This command is a variation of Twist Screen Standard. The command aligns a selected line, polyline or text to be parallel to the east-west direction of your graphics screen.

Think of what you select as a pointer or arrow that will be moved to point in the east direction of the screen. Select the line, polyline or text closest to the endpoint that you want it to point in the horizontal or east direction of the screen.

Prerequisite: None

Keyboard Command: TWIST2

Twist Screen Surveyor

This command is another variation of Twist Screen Standard. You enter the angle/azimuth that you want to be aligned parallel to the east-west direction of the graphics screen.

Prerequisite: None

Keyboard Command: TWIST3

Restore Due North

This command twists the screen to make due north vertical.

Pulldown Menu Location: View > Twist Screen
Keyboard Command: twist4
Prerequisite: None
File Name: \lsp\surv1.lsp

Display Order

This command allows you to change the display order of objects by repositioning an entity from either the background to the forefront of the drawing view or from the forefront to the background of the drawing view.

Prerequisite: None

Keyboard Command: draworder

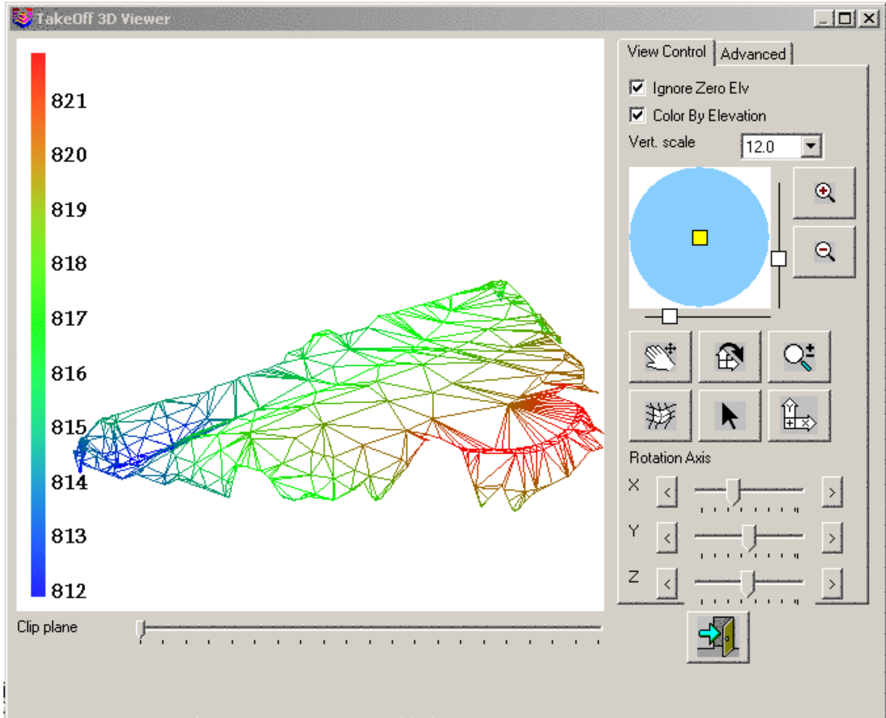
Update Colors For Set Elevations

This command refreshes the color of entities depending on their elevation and layer target. For entities assigned to the Existing or Design layer targets, if the entities are at zero elevation then their color is set to grey. Otherwise the entities have their true, original color. If the Automatic Update Colors command under Settings->Configure->Takeoff Module is toggled off, then this command is the way to update the entity colors after editing elevations.

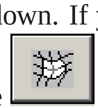
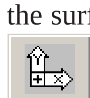
Prerequisites: none
Keyboard Command: update_tk_colors

Existing Surface 3D Viewer

This command allows you to view the existing surface in 3D mode.



In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and the minus magnify glass to zoom out. With the  icon click and drag up to zoom in and drag down to zoom out.The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you want to move. The  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis. The  icon toggles on and off the shading of the surface. The arrow icon reports the elevations at the bottom of the screen as you move around the surface. The  icon restores the surface viewpoint to flat. The  icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prompts

Loading entities...
Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

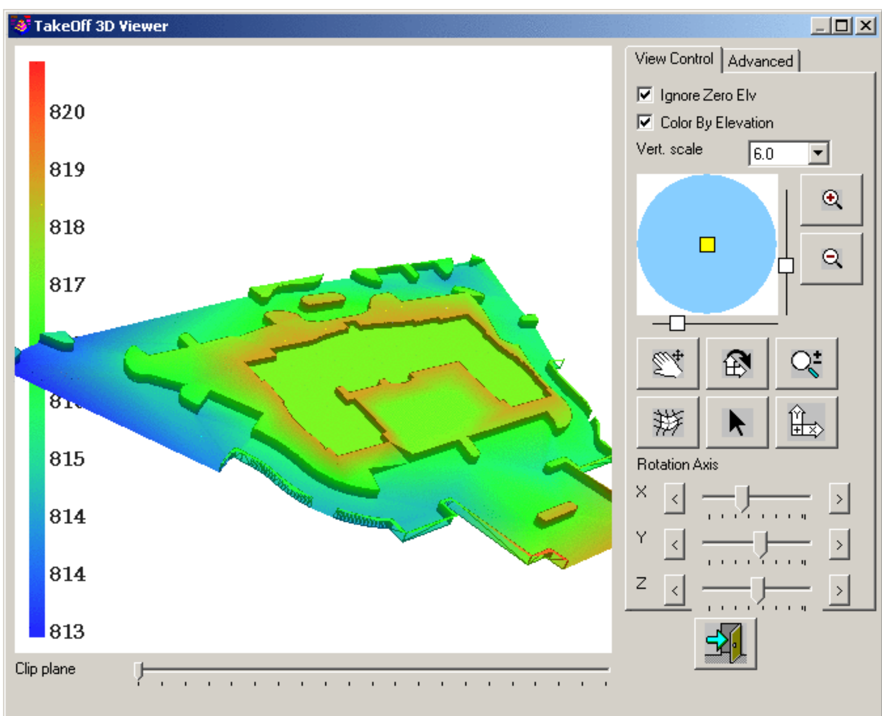
Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

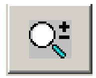
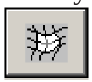
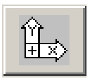
Prerequisite: an existing surface
Keyboard Command: cube_exist

Design Surface 3D Viewer

This command allows you to view the design surface in 3D mode.



In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and the minus magnify glass to zoom out. With the  icon click and drag up to zoom in and drag down to zoom out. The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you want to move. The  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis. The  icon toggles on and off the shading of the surface (the shading is shown in the above drawing). The arrow icon reports the elevations at the bottom of the screen as you move around the surface. The  icon restores the surface viewpoint to flat. The  icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

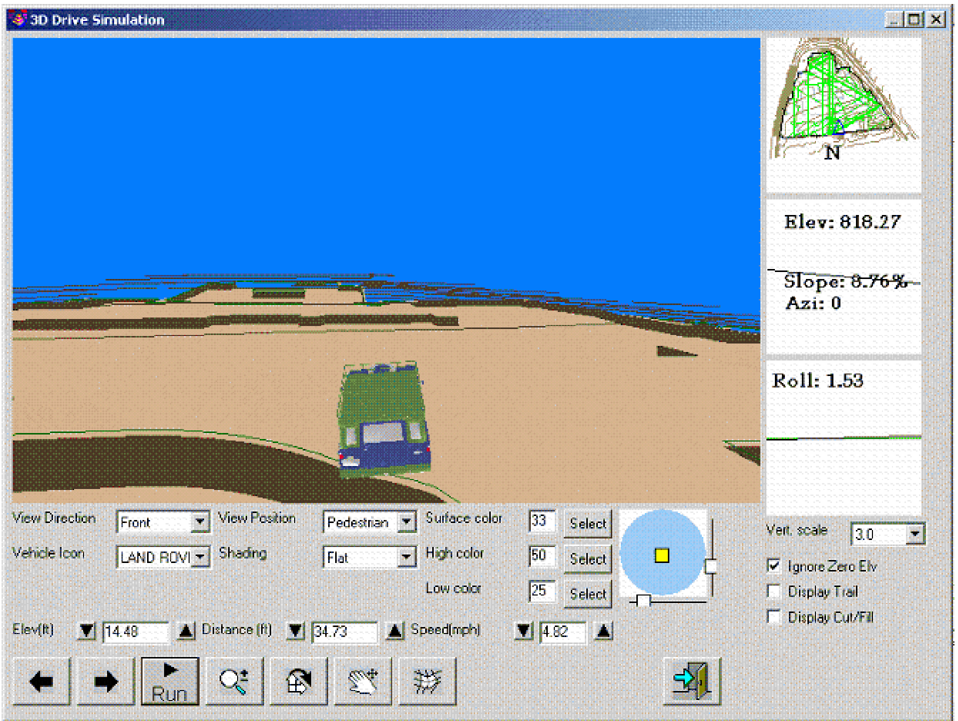
Prompts

Loading entities...
Loading edges...
Loaded 5057 points and 14923 edges

Prerequisite: a design surface
Keyboard Command: cube_design

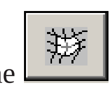
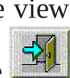
3D Drive Simulation

This command allows you to view and move around the design surface in 3D mode.



Use the arrows on your keypad to move around the drawing.

At the very bottom of the window you will find the basic commands: Run will start to drive your vehicle around the surface, once your vehicle is moving the Run button turns into the Stop button. The arrows moves your vehicle left and right. The magnify glass zooms in and out. Click and drag up to zoom in and click and drag down to zoom out. When your vehicle is stopped the  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis.

The hand icon allows you to pan around the viewer. Click and drag the direction you want to move. The  icon toggles the shading of the surfaces. The  icon exits 3D Driver Simulation.

Above the basic command buttons you can change the Elevation and Distance away from your vehicle. Also, you can set the speed at which your vehicle travels. For a smaller drawing you may want to move around slower, for a larger drawing faster. Note: Unrealistic speeds such as 500 mph in a dozer may cause 3D Drive Simulation to freeze.

View Direction: Sets the direction of the view from the Front, Back, Left, or Right.
Vehicle Icon: You can select which Vehicle you want to use whether: Dozer, Hummer, School Bus or none at all.
View Position: Sets the elevation and distance to either that of the driver, a pedestrian, or bird.
Shading: Here you can set the shading of the surface to either Flat, Smooth, Elevation, Cut/Fill, or none. Flat just shades the contours as the are. Smooth smooths contours to look for realistic. Elevation colors different elevations in different colors so differences can visual be seen. Cut/Fill colors areas of cut differently than areas of fill so they can be visually seen. None merely shows the triangulation.and does not shade in a surface.

You can select the Surface, High, and Low color by enter in an AutoCAD defined color number or you can choose Select to pick a color. The circle on the right determines the shade of the color.

In the top right of the 3D viewer is an aerial map of your surface. Below that the Elevation, Slope percentage, Azimuth, and Roll are updated as your vehicle moves around the surface. Slope and Roll are shown visually here as well.

On the bottom right you can set the Vertical Scale and check to Ignore Zero Elev, Display Trail, and Display Cut/Fill. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Display Trail draws a line where your vehicle has driven. Display Cut/Fill displays the cut and the fill.

Prompts

Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles
Loading reference file...

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Loading entities...
Loading vehicle icon file...

Loading edges...
Loaded 926 points and 2150 edges

Prerequisite: a design surface
Keyboard Command: tk_flyby

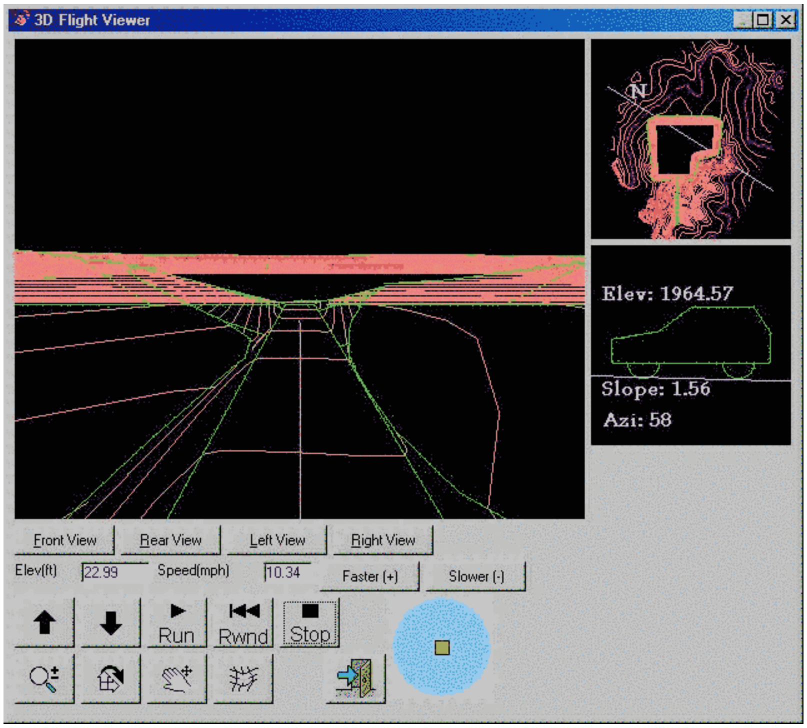
FlyOver Along 3D Polyline

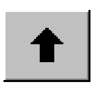
This command allows you to view a self guided animation of following a path through a 3D surface model. There are two variations to this command. When the command is started, you must specify whether you want to use a surface model from file or screen entities.

Surface model from file: Using this method, you can select either a triangulation (.TIN) file or a grid (.GRD) file, then you have the option of following a polyline or following a "free" path. If you choose the polyline method, then the animation is limited to following the polyline. If you choose the "free" path method, you first specify two points to obtain a starting direction, the while inside the viewer you can point the animation in any direction.

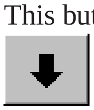
Screen entities: Using this method, you must select a 3D polyline to follow. The animation is limited to following the polyline.

After making the above selections, the 3D graphics window is opened. The main window is for the animation, the smaller upper right window shows you the overall plan view, and the smaller window located at middle right shows you the current elevation, slope and azimuth. While following a "free" path, you will have a 3rd small window located at lower right which shows you the amount of roll at your current position.

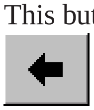




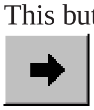
This button raises the elevation of your viewing position.



This button lowers the elevation of your viewing position.

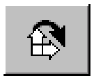


This button turns your viewing position to the left.

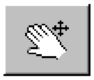


This button turns your viewing position to the right.

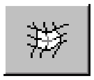
This button allows you to zoom in and out.



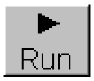
This button allows you to rotate the main animation window in any X, Y or Z direction.



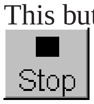
This button allows you to pan.



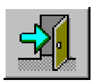
This button toggles shading on and off.



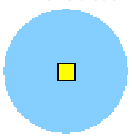
This button starts the animation in the main window.



This button stops the animation.



This button exits the 3D Surface FlyOver command

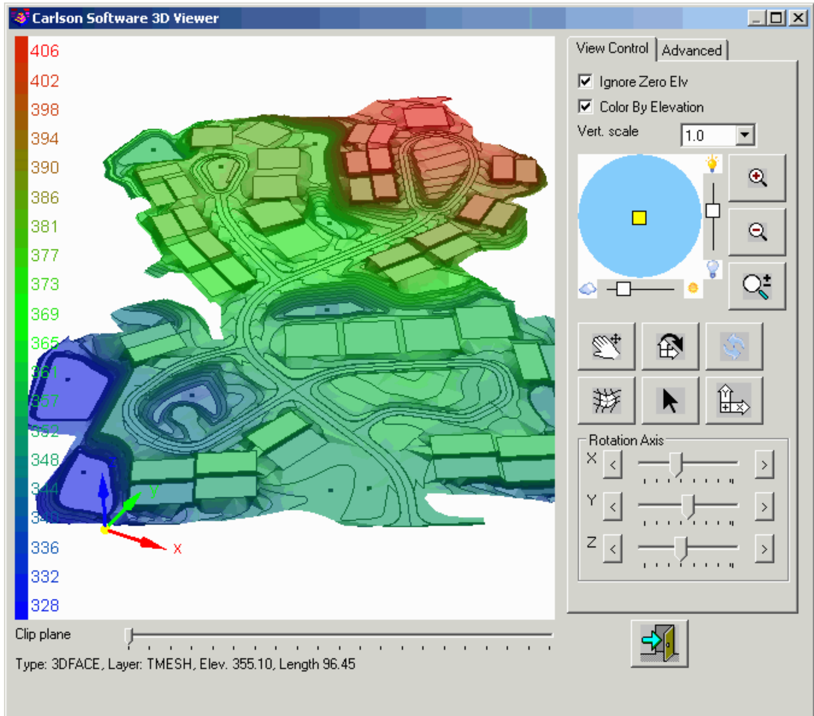


Control for position of the light source, viewed from above.

Prerequisite: Surface Model and optionally a 3D Polyline
Keyboard Command: flyby

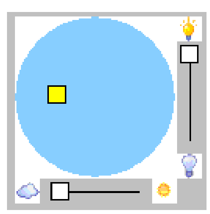
3D Viewer Window

This command views in 3D, the selected 3D faces, polylines, lines and points. This routine uses the OpenGL graphics library for rendering, which gives it superior performance. Some of it's features include the ability to zoom in and out, pan, rotate around the X,Y,Z axis and shade in user-positioned lighting.

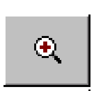


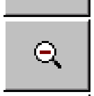
View Control

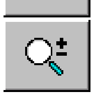
- **Ignore Zero Elevations:** When checked, the 3D viewer ignore entities at zero elevation.
- **Color By Elevation:** This will color the contours or 3D faces by elevation. The elevation scale legend is displayed on the left of the window.
- **Vert. scale:** Sets the vertical scale factor for the 3D viewer. Flat surfaces can be exaggerated by increasing the vertical scale.

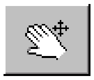


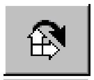
This control represents position of the sun in the sky if looked from above. Therefore, the position of the sun in the center means that the sun is in a zenith, and position near the edge of the circle means that the sun is near the horizon. To move the sun, simply drag it to a new location, or click on the new location. The slide bars on the sides are the intensity and brightness of the display.

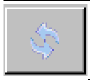
 Zooms IN.

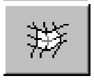
 Zooms OUT.

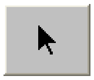
 Switch to Dynamic Zoom mode.

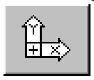
 Switch to Pan mode. Click and drag to pan.

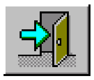
 Switch to Rotation mode.

 Switch to initial view.

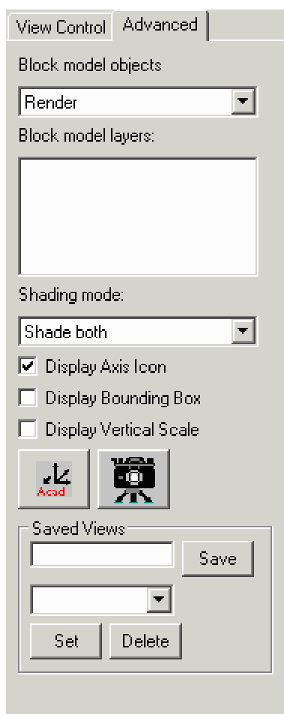
 Toggles shading on and off.

 This is an inquire tool. Point the arrow to any entity to display entity data including the layer, type, elevation and length.

 Resets the 3D view to plan.

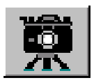
 Exit the 3D viewer window.

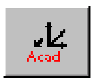
- **Clip Plane:** This slider will clip the image based on the location of the slider. When the slider is all the way to the left, the entire image is displayed. Moving the slider to the right will clip the image, going deeper as the slider is moved to the right. This is useful to view items that are hidden behind something else.
- **Scroll Bars:** Use X,Y,Z scrollbars near the bottom to rotate the view. The range of these scrollbars is -180 to +180 degrees with middle being 0 which is the default position when the viewer starts. When the cursor is near the middle of the window, the XY icon will allow for rotating the image with the mouse, while holding the left mouse button. Move the cursor to the edge, and the icon switches to Z. This allows for rotating around the Z axis with the mouse, while holding the left mouse button.



Advanced Tab

- **Block Model Objects:** This option has three choices when loading block model entities. 1. To leave as points. 2. To Render and 3. To prompt each time. If render is selected, it will apply to all face objects such as a TIN or GRD.
- **Block Model layers:** This will display the block color scheme. Colors of the blocks can be turned on or off to view blocks in the middle.
- **Shading Mode:** There are 3 shading modes to render 3D faces. They are 1. Shade Front, 2. Shade Both, and 3. Shade Back. This will render the top and bottom of the faces if desired.
- **Display Axis Icon:** This controls whether to show the X/Y/Z axis icon in the lower left of the graphic window.
- **Display Bounding Box:** This controls whether to display a 3D box around the limits of the data.
- **Display Vertical Scale:** This controls whether to display the current vertical scale in the graphic window.


This function exports the graphic display to an image file. Several different image file formats are supported including bmp, png, jpg, xpm and gif. There is a Export Image Selections dialog to choose the image resolution and color depth.



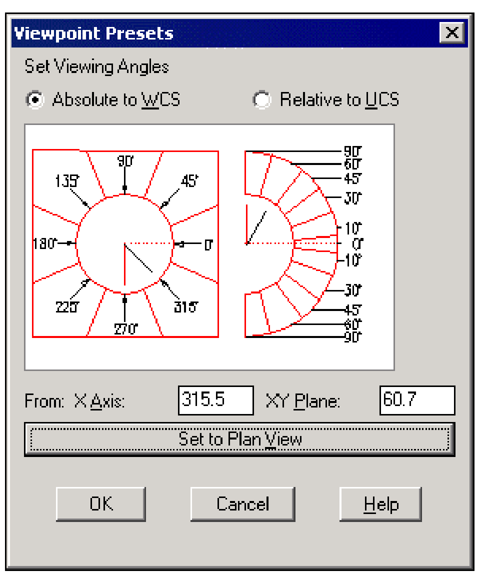
Sets the AutoCAD view to match the view shown in the 3D viewer window.

- **Saved Views:** This option allows for naming and saving a 3D view. These can be selected from the pulldown. They can be deleted from the list.

Pulldown Menu Location: View
Keyboard Command: cube
Prerequisite: Entities to display
File Name: \lsp\cube.arx

Viewpoint 3D

This command allows you to define 3D view settings.



1 Under Set Viewing Angles, you must set the direction of the view relative to either the world coordinate system (WCS) or a user coordinate system (UCS).

- **Absolute to WCS**: This option sets the view direction relative to the WCS.
- **Relative to UCS**: This option sets the view direction relative to the current UCS.

2 You must specify the viewing angles.

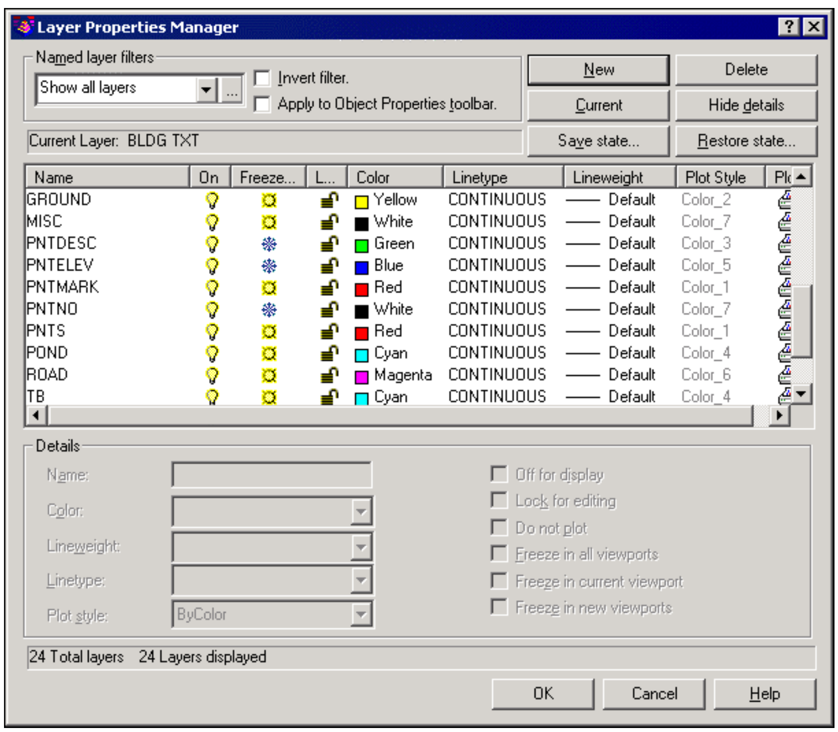
- **X Axis**: This field specifies the angle from the X axis.
- **XY Plane**: This field specifies the angle from the XY plane. You can also use the sample image to specify viewing angles. The black arm indicates the new angle. The red arm indicates the current angle. Specify an angle by selecting the inner region of the circle or half-circle. Selecting the bounded outer regions rounds off the angle to the value displayed in that region.
- **Set to Plan View**: This option sets the viewing angles to display the plan view relative to the selected coordinate system.

Prerequisite: None.

Keyboard Command: DDVPOINT

Layer Control

This command allows you to manage layers and layer properties.



This Layer Properties Manager dialog box makes a layer current, adds new layers to the layer name list, and renames an existing layer. You can assign properties to layers, turn layers on and off, freeze and thaw layers globally or by viewport, lock and unlock layers, set plot styles for layers, and turn plotting on and off for layers. You can filter the layer names displayed in the Layer Properties Manager, and you can save and restore layer states and properties settings.

1 Under Named Layer Filters, you determine which layers to display in the list of layers. You can filter layers based on whether they're xref-dependent, or whether they contain objects. You can also filter layers based on name, visibility, color, linetype, lineweight, plot style name, whether they are plotted, or whether they are frozen in the current viewport or in new viewports.

• **[...]**: This button displays the Named Layer Filters dialog box.

• **Invert Filter**: This option displays layers based on the opposites of the criteria you select when you are using a named layer filter. Layers that fit the inverse criteria are displayed in the layer name list.

• **Apply to Object Properties Toolbar**: This option displays in the Object Properties toolbar only layers that match the current filter. The layer list tooltip on the Object Properties toolbar displays the filter status of layers in the drawing. (To display the layer list tooltip, position the pointing device over the layer list on the Object Properties toolbar.)

• **New**: This option creates a new layer. After you choose New, the list displays a layer named LAYER1. You can edit this layer immediately. To create multiple layers quickly, you can select a layer name for editing and enter multiple layer names separated by commas. If you create a new layer, the new layer inherits the properties of the currently selected layer in the layer list (such as Color, and On/Off state). To create layers with default settings, make sure that there are no selected layers in the list or that you select a layer with default settings before beginning layer creation.

• **Current**: This option sets the selected layer as the current layer. The CLAYER system variable stores the layer name.

• **Delete**: This option deletes selected layers from the drawing file definition. You can delete only unreferenced layers. Referenced layers include layers 0 and DEFPOINTS, layers containing objects (including objects in block definitions), the current layer, and xref-dependent layers. Layers that don't contain objects (including objects in block definitions), are not current, and are not xref-dependent can be deleted by using the PURGE command. Be careful about deleting layers if you are working on a drawing in a shared project or one based on a set of layering standards.

• **Show/Hide Details**: This option controls whether the Details section is displayed in the Layer Properties Manager.

• **Save State**: This option displays the Save Layer States dialog box, in which you save layer state and layer properties settings of all layers in a drawing. You can choose which layer states and properties you want to preserve. You save a layer state by assigning it a name.

• **Restore State**: This option displays the Layer States Manager, in which you can manage named layer states.

2 The Layer Properties Manager dialog box displays all layers and their properties. To modify a property, click its icon. To quickly select all layers, right-click your pointing device and use the shortcut menu. The following are the layer properties you can modify:

• **Name**: This field displays the names of the layers. You can select a name, and then click and enter a new name.

• **On/Off**: This field turns layers on and off. When a layer is on, it is visible and available for plotting. When a layer is off, it is invisible and not plotted, even if Plot is on.

• **Freeze/Thaw in All Viewports**: This field freezes and thaws layers in all floating viewports. A frozen layer is invisible and excluded from regeneration, hiding objects, rendering, and plotting. A thawed layer is visible and available for regeneration, hiding objects, rendering, and plotting.

You can freeze layers to speed up ZOOM, PAN, and many other operations, improve object selection performance, and reduce regeneration time for complex drawings. TakeOff does not display, plot, or regenerate objects on frozen layers. Objects on frozen layers do not hide objects and are not rendered.

You can freeze layers in all viewports, in the current viewport, or in new viewports.

Freeze layers that you want to be invisible for long periods. When you thaw a frozen layer, the program regenerates and displays the objects on that layer. If you switch between visible and invisible states frequently, use the On/Off setting.

- **Lock/Unlock**: This field locks and unlocks the layers. You cannot select or edit objects on a locked layer. Locking a layer is useful if you want to view information on a layer for reference but do not want to edit objects on that layer.

- **Color**: This field changes the color associated with the selected layers. Clicking the color name displays the Select Color dialog box.

- **Linetype**: This field changes the linetype associated with the selected layers. Clicking any linetype name displays the Select Linetype dialog box.

- **Lineweight**: This field changes the lineweight associated with the selected layers. Clicking any lineweight name displays the Lineweight dialog box.

- **Plot Style**: This field changes the plot style associated with the selected layers. If you are working with color-dependent plot styles (the PSTYLEPOLICY system variable is set to 1), you cannot change the plot style associated with a layer. Clicking any plot style displays the Select Plot Style dialog box.

- **Plot/Don't Plot**: This field controls whether the selected layers are plotted. If you turn off plotting for a layer, the objects on that layer are still displayed. Turning off plotting for a layer affects only visible layers in the drawing (layers that are on and thawed). If a layer is set to plot, but is currently frozen or off in the drawing, TakeOff does not plot the layer. Turning off plotting for layers containing reference information such as construction lines can be useful.

Prerequisite: None

Keyboard Command: LAYER

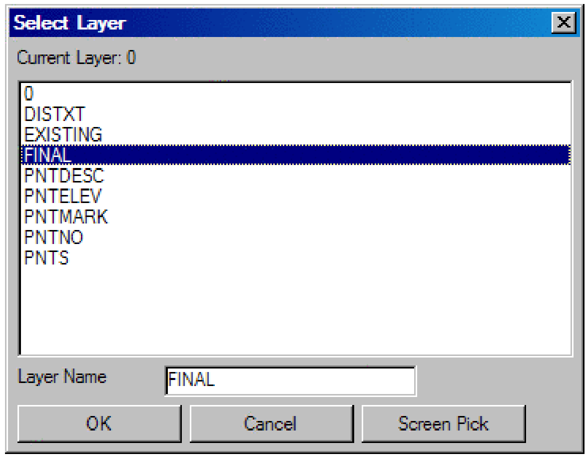
Set Layer

This command allows the user to change the current layer to a different layer by picking an entity on that layer.

Pulldown Menu Location: View
Keyboard Command: lset
Prerequisite: None
File Name: \lsp\picklayr.lsp

Change Layer

This command allows you to change the layer of a group of entities by selecting the group of entities. The layer name to assign can be either typed it or read from an existing entity by picking an entity that is on the layer that you want to change the group to.



Prompts

Select entities to be changed.
Select objects: *pick entities*
The Select Layer dialog appears *select a layer from the list, or select Screen Pick*
If Screen Pick is chosen,
Pick entity with layer to change to: *pick another entity* This assigns the selected entities to the layer of this entity.
or
Enter new layer name or pick entity with layer (Enter/<Pick>)? *E*
Enter new layer name: *FINAL* This assigns the selected entities to the FINAL layer.

Pulldown Menu Location: View
Keyboard Command: lchg
Prerequisite: None
File Name: \lsp\chglayr.lsp

Freeze Layer

This command will freeze layers by picking entities on that layer.

Pulldown Menu Location: View
Keyboard Command: loff
Prerequisite: None
File Name: \lsp\loff.lsp

Thaw Layer

This command thaws the layers frozen by the Freeze Layer command.

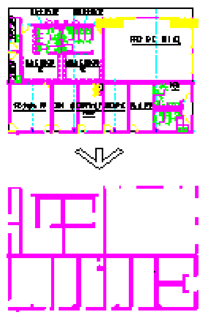
Pulldown Menu Location: View
Keyboard Command: lon
Prerequisite: None
File Name: \lsp\lon.lsp

Isolate Layer

This command freezes all the layers except the ones you select an entity on. The program prompts to see if you would like to retain the POINT layers which keeps the Carlson point layers from freezing. By default, these layers include PNTNO, PNTMARK, PNTDESC, and PNTELEV.

Prompts

Select objects on layers to isolate.
Select objects: *pick entities*
Retain POINT layers [Yes/<No>]? *Press Enter*



Isolate the wall layer by picking one wall line

Pulldown Menu Location: View
Keyboard Command: isolate
Prerequisite: None
File Name: \lsp\isolate.lsp

Restore Layer

This command thaws the layers that were frozen by the *Isolate Layer* command.

Pulldown Menu Location: View
Keyboard Command: restore
Prerequisite: None
File Name: \lsp\restore.lsp

Draw Menu 5

Line

This command allows you to draw a line entity by picking points on the screen or by supplying the coordinate values using the point number and associated coordinates stored in the current coordinate file. The Line command links the line with the points when the line is drawn using point numbers if the Link Linework with Points option is turned on. This option is set under General Settings in the Configure command in the Settings menu. With links active, changing a point with a command like Move Points automatically updates the line. This command always draws 2D lines with a zero elevation.

Prompts

1 Pick point or point numbers: 1-3

You may enter a single point number or a range of point numbers

2 Undo/Distance/<Pick point or point numbers>: 16

3 Undo/+/-/Close/Distance/<Pick point or point numbers>: 35

4 Undo/+/-/Close/Distance/<Pick point or point numbers>: +

The + or **-** activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line.

5 Perpendicular Distance Right: 80

6 Undo/+/-/Close/Distance/<Pick point or point numbers>: **-**

The + or **-** activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line.

7 Perpendicular Distance Left: 105.12

8 Undo/+/-/Close/Distance/<Pick point or point numbers>: D

The distance option allows you to input a distance for the next line segment. The position of the cursor determines the angle.

9 Enter distance: 174.32

10Undo/+/**-**/Close/Distance/<Pick point or point numbers>: C

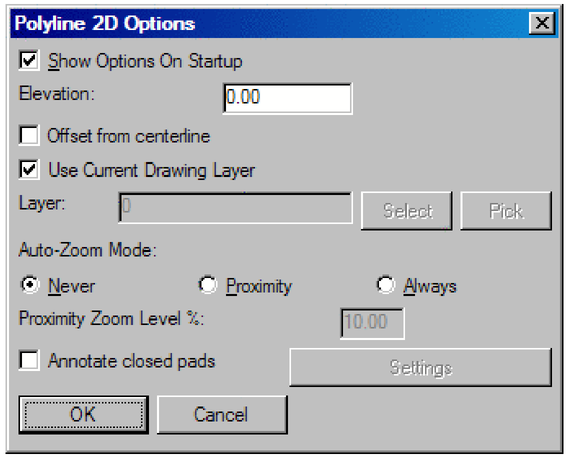
The close option draws a line segment back to the original starting point

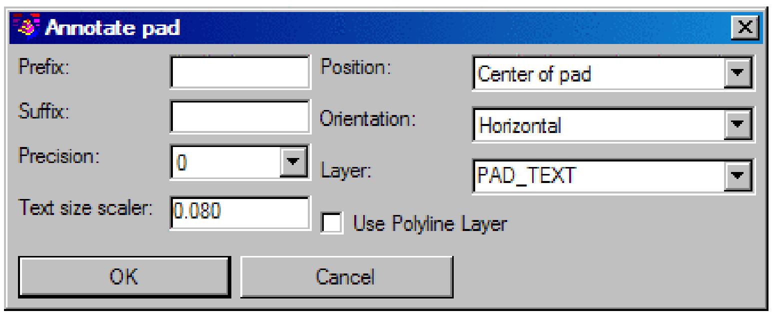
Prerequisite: None

Keyboard Command: 2DLINE

2D Polyline

A polyline is a series of line and/or arc segments joined together in one entity or object. In addition to the regular AutoCAD Draw Polyline command (PL), there is this Carlson command that has several key features and options. This command allows you to draw a polyline entity in a variety of ways, including: Continue, Extend, Follow, Arc, Direction, Close, Pick point or point numbers. This dialog shown below automatically appears when you run the command.





The **Show Options on Startup** dialog will appear every time the command is run, unless this is turned off. If you do not want it to come up, then toggle it off. In this dialog you can set the elevation of the polyline, as well as the layer it will appear on.

Offset from centerline allows you to make another choice. First, a unique command line appears.

[Continue/Extend/Follow/**Offset**/Options/<Pick point or point numbers>]:

Auto-Zoom mode regenerates the screen as you draw your polyline, to give you the best view as you are working. **Always** means it will always regenerate the screen. **Proximity** regenerates the screen according to the **Proximity Zoom Level %** value. **Never** toggles off this feature. The **Elevation** of the polyline can be set here. The default is 0. If it is off, then the last settings will apply. To get the box back, choose O for Options on the command line.

If **Use Current Drawing Layer** is on, the layer of the new polyline will be the current layer. If the current layer is not used, the **Layer** option allows you to **Select** from a list or **Pick** from the screen.

There are three options under **Auto-Zoom Mode**. **Never** will not zoom to the last point picked. **Proximity** will zoom to the percent proximity set below. **Always** will always zoom center on every point.

If the **Proximity** Auto-Zoom mode is checked, the percent of the proximity is set in the box on the right.

Annotate closed pads, when checked, will make available the Settings button on the right side. When Settings is chosen, the Annotate pad dialog appears. Here is where you can enter in values that will result in annotation for the closed polyline.

With regards the options available on the command line, they are explained as follows:

Continue allows you to run Arc, Direction, Close, Extend, or Follow for an existing polyline.

Extend allows you to extend an existing polyline by picking or entering in a distance.

Follow allows you to match the path of your polyline with an existing polyline. You are prompted to select the point in which your polyline first intersects with the existing polyline and the point in which it exits. If the existing polyline is a closed polyline than you will be prompted to approve the direction in which your polyline follows the existing closed polyline.

Options brings up the Polyline 2D Options dialog that appears when you first run the command, unless Offset from centerline is checked, in which case the letter indicates this feature, and a centerline will be requested.

Pick point allows you to pick a point on the screen to start your polyline.

Point numbers allows you to enter in coordinates to start your polyline.

Arc allows you to draw an arc by Arc length/Chord/Radius/Second point in your polyline. The + or **-** activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line. This is useful for plotting buildings.

Distance for an angle code, a backsight point, an angle (dd.mmss), and a distance to draw your polyline.

Close will connect the ends of your polyline making it a closed polyline.

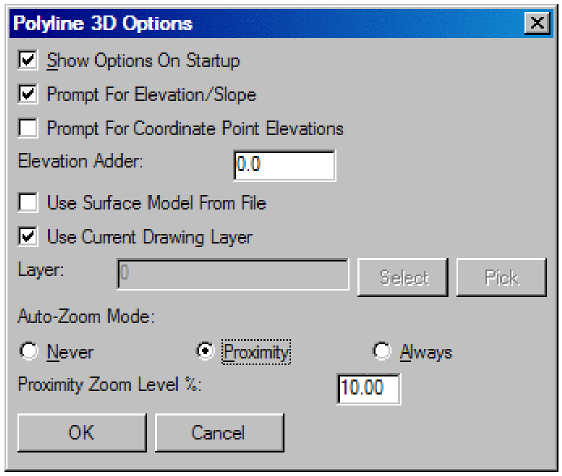
Prompts

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: *pick a point*
Segment length: 0.00, Total length: 0.00
[Arc/Close/Direction/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *pick a point*
Segment length: 3.83, Total length: 3.83
[Arc/Close/Direction/Extend/Follow/Line/Undo/<Pick point or point numbers>]: *pick a point*
Segment length: 2.94, Total length: 6.77 *press Enter to end*

Pulldown Menu Location: Draw
Keyboard Command: 2DP
Prerequisite: None
File Names: \lsp\poly3d.arx

3D Polyline

In addition to the regular AutoCAD Draw 3D Polyline command, there is this Carlson command that has several options.



The **Show Options on Startup** dialog will appear every time the command is run, unless this is turned off. If it is off, then the last settings will apply. To get the box back, choose O for Options.

Prompt for Elevation/Slope controls whether the elevation of each picked point will be entered in, or hit S for slope to draw a slope line.

Use Surface Model from File will use a grid or triangulation file as a surface model. Wherever the points are picked on the surface, the elevation of the surface will be assigned to the polyline.

If **Use Current Drawing Layer** is on, the layer of the new polyline will be the current layer.

If the current layer is not used, the **Layer** option allows you to **Select** from a list or **Pick** from the screen.

There are 3 options under **Auto-Zoom Mode.** Never will not zoom to the last point picked. Proximity will zoom to the percent proximity set below. Always will always zoom center on every point.

If the Proximity Auto-Zoom mode is checked, the percent of the proximity is set in the **Proximity Zoom Level%** box.

Prompts

[Continue/Extend/Follow/Options/<Pick point or point numbers>]: *pick a point*
Elevation <0.00>: *435*
Z: 435.00, Hz dist: 0.00, Slope dist: 0.00, Slope: 0.0% Ratio: 0.0:1
[Arc/Direction/Close/Follow/Undo/<Pick point or point numbers>]: *pick a point*
Slope/Ratio/Interpolate/Degree/<Elevation> <0.00>: *444*
Z: 444.00, Hz dist: 3.67, Slope dist: 9.72, Slope: 245.3% Ratio: 0.4:1
[Arc/Direction/Close/Extend/Follow/Undo/<Pick point or point numbers>]: *pick a point*
Slope/Ratio/Interpolate/Degree/<Elevation> <0.00>: *399*
Z: 399.00, Hz dist: 3.16, Slope dist: 45.11, Slope: -1425.2% Ratio: -0.1:1
[Arc/Direction/Close/Extend/Follow/Undo/<Pick point or point numbers>]: *press Enter to end*

Pulldown Menu Location: Draw
Keyboard Command: 3DP
Prerequisite: None

Circle

This command allows you to draw a circle.

Prompts

1 Pick center point or point number or [3P/2P/TTR]: pick point or specify option

- 3P: This option draws a circle based on three points on the circumference.
- 2P: This option draws a circle based on two endpoints of the diameter.
- TTR-Tangent, Tangent, Radius: This option draws a circle with a specified radius tangent to two objects.

2 Specify radius of circle or [Diameter]: enter a value

Sometimes more than one circle matches the criteria specified in the command. The circle whose tangent points are closest to the selected points is drawn.

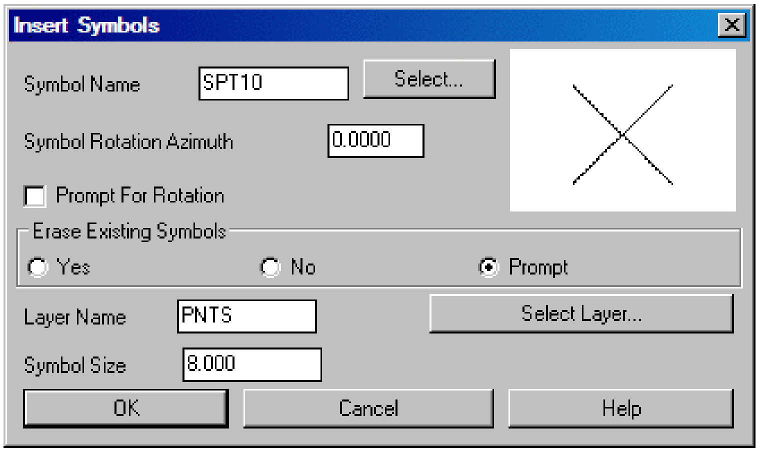
Prerequisite: None

Keyboard Command: SCIRCLE

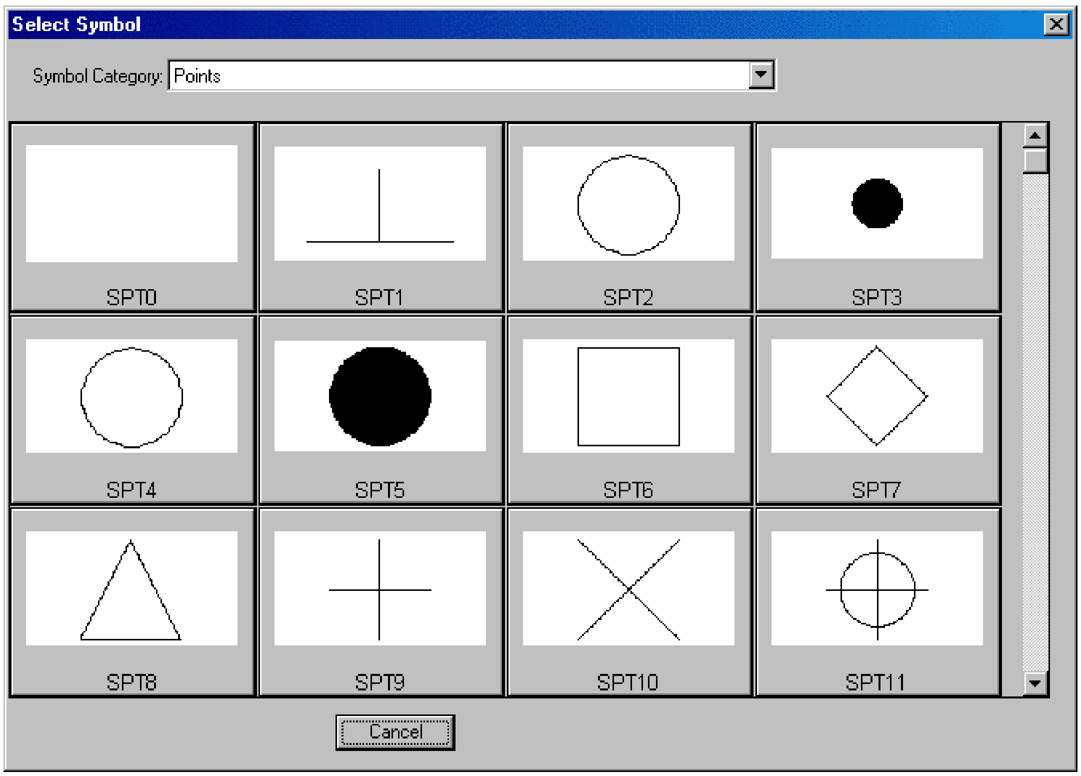
Symbols

Function

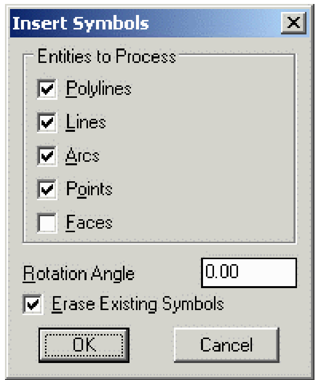
This command inserts symbols from the symbol library into the drawing. The symbol library may be edited using the *Edit Symbol Library* command. The locations for the symbols can be specified by picking points, specifying point numbers in the current coordinate (.CRD) file or by entering the northing and easting. If you specify a point number, and that point number already has a symbol on it, you will be prompted whether or not to replace the existing symbol. Selecting the Enter coords option allows you to insert the symbol by entering a northing, easting and elevation. Using the Select entities option, symbols can also be placed on arcs, faces, points, text, lines and polylines. Under the Options command, you can turn prompting for rotation on or off. With rotation off, the symbol will be inserted horizontal to the current twist screen. Choose a symbol from the Select Symbol dialog by clicking on it. The Symbol Category choices are Points, Trees and Map Symbols. You may select a category by choosing the Symbol Category dropdown list. Within each category, use the scroll bar to view all of the symbols.



Appears at start of command



Appears when Select (symbol) is chosen



Select entities dialog box

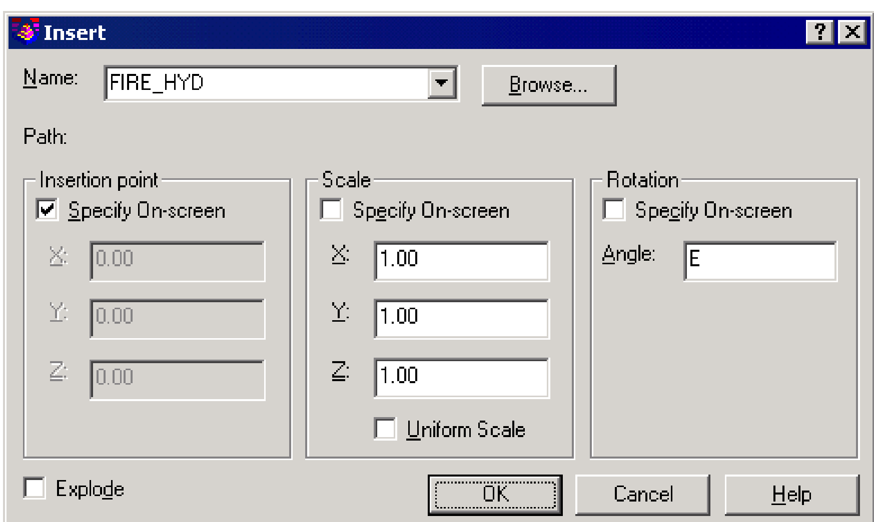
Prompts

Insert Symbols dialog *choose variables and click OK*
Options/Select entities/Enter coords/<Pick point or point numbers>: *pick a point*
Options/Select entities/Enter coords/<Pick point or point numbers>: *5-10* Inserts symbols at points 5-10 from the current coordinate file.
Options/Select entities/Enter coords/<Pick point or point numbers>: *S*
Insert Symbols dialog
Select arcs, faces, points, text, lines and polylines. *select objects*
Options/Select entities/Enter coords/<Pick point or point numbers>: *press Enter to end*

Keyboard Command: ptsym
Prerequisite: None

Insert Drawing

This command allows you to place a named block or drawing into the current drawing.



1 In the Insert dialog box, you specify the block to insert and define the position for the inserted block. The last block you insert during the current editing session becomes the default block for subsequent uses of this command.

- Name: This field specifies the name of a block to insert or the name of a file to insert as a block.
- Browse: This button opens the Select Drawing File dialog box (a standard file selection dialog box) where you can select a block or a file to insert.

2 Under Insertion Point, you specify the insertion point for the block.

- Specify On-Screen: This option specifies the insertion point of the block using the pointing device.
- X: This field sets the X coordinate value.
- Y: This field sets the Y coordinate value.
- Z: This field sets the Z coordinate value.

3 Under Scale, you specify the scale for the inserted block. Specifying negative values for the X, Y, and Z scale factors inserts a mirror image of a block.

- Specify On-Screen: This option specifies the insertion point of the block using the pointing device.
- X: This field sets the X coordinate value.
- Y: This field sets the Y coordinate value.
- Z: This field sets the Z coordinate value.
- Uniform Scale: This option specifies a single scale value for X, Y, and Z coordinates. A value specified for X is also reflected in the Y and Z values.

4 Under Rotation, you specify the rotation angle for the inserted block.

- Specify On-Screen: This option specifies the rotation angle of the block using the pointing device.
- Angle: This field sets a rotation angle for the inserted block.

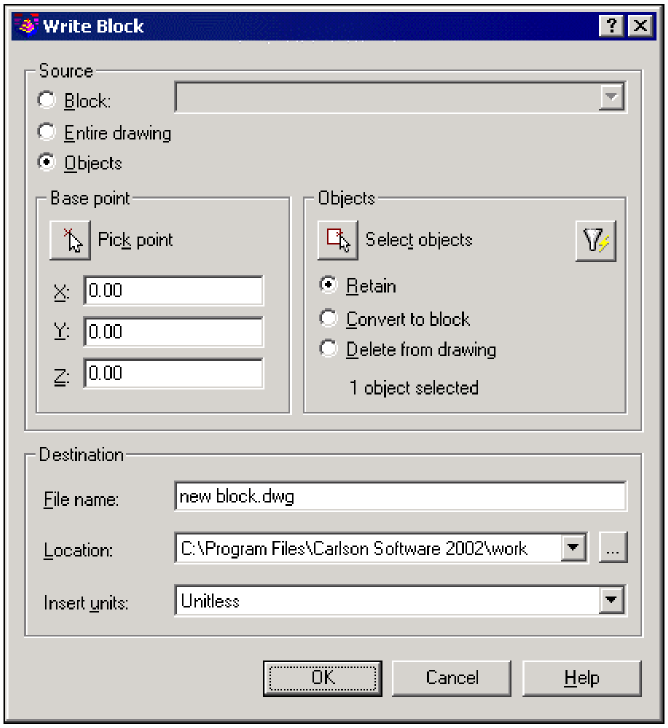
5 You can explode the block and inserts to the individual parts of the block. When you select Explode, you specify only an X scale factor.

Prerequisite: None

Keyboard Command: DDINSERT

Write Block

This command allows you to write objects or a block to a new drawing file.



The Write Block dialog box displays different default settings depending on whether nothing is selected, a single block is selected, or objects other than blocks are selected. For example, if you have a single block selected when you open the Write Block dialog box, the Source radio button is set to Block.

1 Under Source, you write selected blocks and objects out as a file, and specify insertion points.

- Block: This option specifies an existing block to save as a file. Select a name from the list.
- Entire Drawing: This option selects the current drawing as a block.
- Objects: This option specifies objects to be saved as a file.

2 Under Base Point, you must specify a base point for the block. The default value is 0,0,0.

- X: This field specifies the X coordinate value.
- Y: This field specifies the Y coordinate value.
- Z: This field specifies the Z coordinate value.
- Pick Point: This option allows you to temporarily close the dialog box so that you can specify an insertion base point in the current drawing.

3 Under Objects, you specify the objects to include in the new block and whether to retain or delete the selected objects or convert them to a block instance after you create the block.

- Retain: This option retains the selected objects as distinct objects in the drawing after you create the block.
- Convert to block: This option converts the selected objects to a block instance in the drawing after you create the block.
- Delete from drawing: This option deletes the selected objects from the drawing after you create the block.
- Select objects: This option dismisses the Block Definition dialog box temporarily while you select the objects for the block. When you finish selecting objects, press Enter to redisplay the Block Definition dialog box.
- Quick Select: This option displays the Quick Select dialog box, which defines a selection set.

- Objects Selected: This option displays the number of selected objects.

4 Under Destination, specify the name, location, and unit value used for the objects in the file.

- File Name: This field specifies a file name that the block or objects will be saved to.
- Location: This field specifies the drive and directory path for the file.
- Insert Units: This field specifies the unit value to be used when the new file is inserted as a block. Enter 0 (zero) if you do not want to scale the drawing to a specific value as you insert it.

Prerequisite: Drawing entities

Keyboard Command: WBLOCK

Text

Creates a single-line text object.

You can use the TEXT to enter several lines of text that you can rotate, justify, and resize. As you type at the Enter Text prompt, the text you are typing is displayed on the screen. Each line of text is a separate object. To end a line and begin another, press Enter after entering characters at the Enter Text prompt. To end the TEXT command, press Enter without entering any characters at the Enter Text prompt.

By applying a style to the text, you can use a variety of character patterns or fonts that you can stretch, compress, make oblique, mirror, or align in a vertical column.

If TEXT was the last command entered, pressing ENTER at the Specify Start Point of Text prompt skips the prompts for height and rotation angle and immediately displays the Enter Text prompt. The text is placed directly beneath the previous line of text. The point specified at the prompt is also stored as the Insertion Point object snap.

Prompts

1 Current text style: "MONO" Text height: 4.00

2 Specify start point of text or [Justify/Style]: S

The style option lets you change the textstyle on the fly

3 Enter style name or [?] <MONO>: STANDARD

4 Current text style: "STANDARD" Text height: 4.00

5 Specify start point of text or [Justify/Style]: J

The justify option lets you specify the justification for the text.

6 Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]: BC

In this case BC = Bottom Center

7 Specify bottom-center point of text: pick point or enter coordinates

8 Specify height <4.00>: press enter to select default or enter text height

9 Specify rotation angle of text <0d0'0">: press enter to select default or enter angle

10Enter text: Found Iron Pin

11Enter text: press enter to end

Prerequisite: None

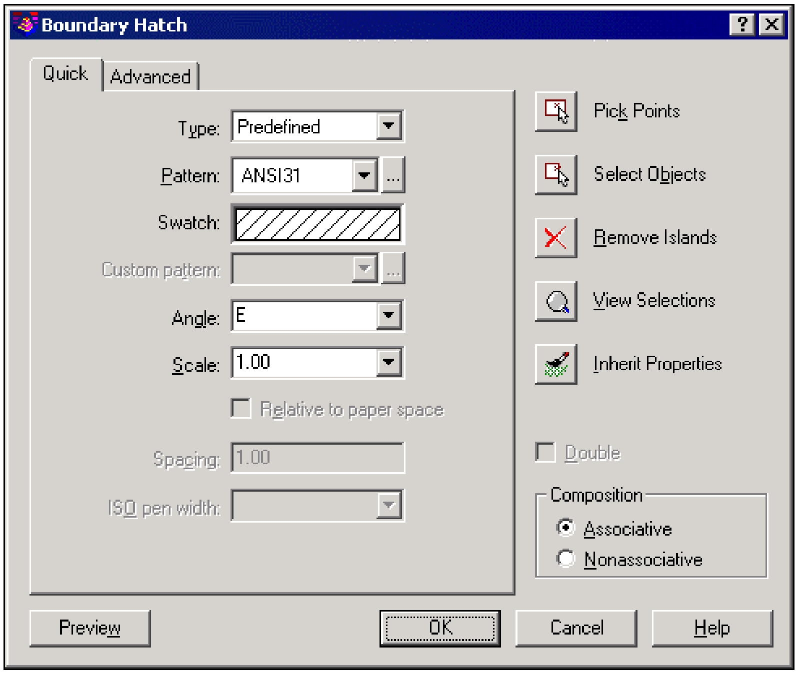
Keyboard Command: DTEXT, TEXT

Hatch

This command allows you to fill an enclosed area or selected objects with a hatch pattern.

The Hatch command first defines the boundaries of the area you want to hatch, either by computing a region or polyline boundary from a specified point within an enclosed area, or by using selected objects as boundaries. It then fills the boundaries with a hatch pattern or a solid color. You can create an associative hatch, which updates when its boundaries are modified, or a nonassociative hatch, which is independent of its boundaries. You can preview any hatch and adjust the definition.

Due to the large number of combinations of geometry that you can hatch, editing hatched geometry can produce unexpected results. In this event, delete the hatch object and rehatch.



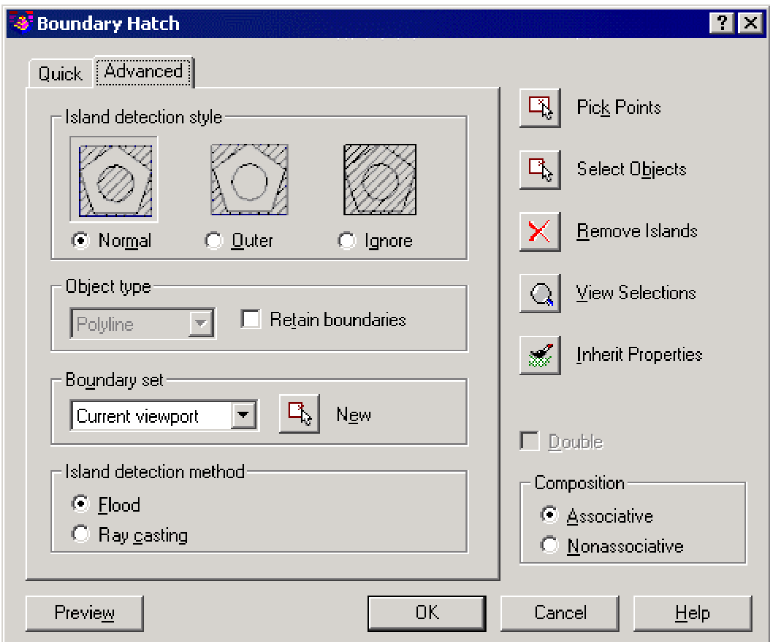
The Boundary Hatch dialog box defines the boundary, pattern type, pattern properties, and attributes for hatch objects. Use the Quick Tab to work with hatch patterns and quickly create a hatch. Use the Advanced Tab to customize how TakeOff creates and hatches boundaries.

1 Under the Quick Tab you define the appearance of the hatch pattern to be applied.

- Type: This field sets the pattern type.
- Pattern: This field lists the available predefined patterns. The six most recently used predefined patterns appear at the top of the list. The Pattern option is available only if you set Type to Predefined.
- [...]: This button displays the Hatch Pattern Palette dialog box, in which you can view preview images for all predefined patterns at once to help you make a selection.
- Swatch: This field displays a preview of the selected pattern. You can click the swatch to display the Hatch Pattern Palette dialog box.
- Custom Pattern: This field lists the available custom patterns. The six most-recently used custom patterns appear at the top of the list. The Custom Pattern option is available only if you set Type to Custom.
- Angle: This field specifies an angle for the hatch pattern relative to the X axis of the current UCS.

- Scale: This option expands or contracts a predefined or custom pattern. This option is available only if you set Type to Predefined or Custom.
- Relative to Paper Space: This option scales the hatch pattern relative to paper space units. Using this option, you can easily display hatch patterns at a scale that is appropriate for your layout. This option is available only from a layout.
- Spacing: This option specifies the spacing of lines in a user-defined pattern. This option is available only if you set Type to User Defined.
- ISO Pen Width: This option scales an ISO predefined pattern based on the pen width you choose. This option is available only if you set Type to Predefined and set Pattern to one of the available ISO patterns.

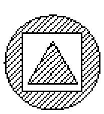
2 Under the Advanced Tab you define how TakeOff creates and hatches boundaries.



- Island Detection Style: This option allows you to specify the method for hatching objects within the outermost hatch boundary. If no internal boundaries exist, specifying an Island Detection style has no effect. Because you can define a precise set of boundaries, it's often best to use the Normal style.

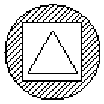
The illustrations that accompany each style show how the program hatches a group of three nested boundary objects in each case.

Normal



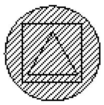
Hatches inward from the outer boundary. If the program encounters an internal intersection, it turns off hatching until it encounters another intersection. Thus, areas separated from the outside of the hatched area by an odd number of intersections are hatched, and areas separated by an even number of intersections are not.

Outer



Hatches inward from the outer boundary. The program turns hatching off if it encounters an internal intersection. Because this process starts from both ends of each hatch line, the program hatches only the outermost level of the structure and leaves the internal structure blank.

Ignore



Ignores all internal objects and hatches through them.

Hatching concave curves with the Outer and Ignore styles can cause hatching discrepancies.

The Normal, Outer, and Ignore options are also available from a shortcut menu by right-clicking in the drawing area while you specify points or select objects to define your boundaries.

• Object Type: This option allows you to specify whether to retain boundaries as objects, and specifies the object type TakeOff applies to those boundary objects. Object Type controls the type of the new boundary object. TakeOff creates the boundary as a region or a polyline. This option is available only if you select Retain Boundaries.

• Retain Boundaries: This option adds the temporary boundary objects to the drawing.

• Boundary Set: This field defines the set of objects TakeOff analyzes when defining a boundary from a specified point. The selected boundary set has no effect when you use Select Objects to define a boundary. By default, when you use Pick Points to define a boundary, the program analyzes all objects visible in the current viewport. By redefining the boundary set, you can disregard certain objects when defining boundaries without having to hide or remove those objects. For large drawings, redefining the boundary set can also produce the boundary faster because the program examines fewer objects.

• New: This option prompts you to select the objects that define the boundary set. When you choose this option, the dialog box temporarily closes, prompting you to select objects. TakeOff includes only the hatchable objects you select when it constructs the new boundary set. TakeOff discards any existing boundary set, replacing it with the new boundary set defined by the objects you select. If you don't select any hatchable objects, the program retains any current set. Until you exit the Hatch command or create a new boundary set, TakeOff ignores objects that do not exist in the boundary set when you define your boundaries using Pick Points.

• Island Detection Method: This option allows you to specify whether to include objects within the outermost boundary as boundary objects. These internal objects are known as islands.

• Flood: This option includes islands as boundary objects.

• Ray Casting: This option runs a line from the point you specify to the nearest object and then traces the boundary in a counterclockwise direction, thus excluding islands as boundary objects.

3 In the Boundary Hatch dialog box, you set the options the define the selection set.

• Pick Points: This option determines a boundary from existing objects that form an enclosed area. How TakeOff detects objects using this option depends on the selected Island Detection Method on the Advanced tab. For example, if the Island Detection Method is Flood, the program detects objects within the outermost boundary as islands and includes them in the boundary definition. The Island Detection Style (which you also set on the Advanced tab) then determines how to hatch the detected islands. When you choose Pick Points, the dialog box closes temporarily, and the program prompts for point specification.

- Select Objects: This option allows you to select specific objects for hatching. The dialog box closes temporarily, and the program prompts you for object selection. When you define your boundaries using Select Objects, the program does not detect interior objects automatically. You must select the objects within the selected boundary to hatch those objects according to the current Island Detection Style (which you set on the Advanced tab). Each time you choose Select Objects, the program clears the previous selection set. While selecting objects, you can right-click at any time in the drawing area to display a shortcut menu. You can undo the last or all selections, change the selection method, change the island detection style, or preview the hatch.
- Remove Islands: This option removes from the boundary definition any of the objects that the program detects as islands when you use Pick Points. You cannot remove the outer boundary.
- View Selections: This option temporarily dismisses the dialog box and displays the currently defined boundaries with the hatch settings that you last previewed. This option is unavailable when you have not yet specified points or selected objects.
- Inherit Properties: This option hatches specified boundaries using the hatch properties of one object. After selecting the associative hatch object whose properties you want the hatch to inherit, you can right-click in the drawing area and use the shortcut menu to toggle between the Select Objects and Pick Internal Point options to create boundaries.
- Double: For user-defined patterns, this option draws a second set of lines positioned at 90 degrees to the original lines, creating a crosshatch. This option is available only if you set Type to User Defined on the Quick tab.
- Associative: This option creates an associative hatch, meaning that the hatch is updated when you modify its boundaries.
- Nonassociative: This option creates a nonassociative hatch, meaning that it is independent of its boundaries.
- Preview: This option temporarily dismisses the dialog box and displays the currently defined boundaries with the current hatch settings. This option is not available when you have not yet specified points or selected objects to define your boundaries.

Prerequisite: None

Keyboard Command: BHATCH

2 Tangents, Radius

This command fits a curve between two tangent lines by entering a known radius. It prompts for the radius and then prompts to pick points on the two tangent lines.

Prompts

Radius of Arc <300.000>: *press Enter*
[nea] Pick Point on 1st Tangent Line:*pick a point*
[nea] Pick Point on 2nd Tangent Line: *pick a point*

Pulldown Menu Location: Draw > Arc
Keyboard Command: 2tanlin
Prerequisite: Tangent lines should be drawn before execution
File Name: \lsp\2tanlin.lsp

2 Tangents, Arc Length

This command fits a curve between two tangent lines and a known arc length. It prompts for the arc length then pick the P.I. (intersection of tangent lines) and points on the two tangent lines.

Prompts

Arc Length <100.00>: *press Enter or enter distance*
[int on] Pick P.I. of curve: *pick intersection of tangent lines*
[nea on] Pick pnt on 1st Tangent Line: *pick a point*
[nea on] Pick pnt on 2nd Tangent Line: *pick a point*

Pulldown Menu Location: Draw > Arc
Keyboard Command: 2tanlal
Prerequisite: Tangent lines should be drawn before execution
File Name: \lsp\2tanlal.lsp

2 Tangents, Chord Length

This command fits a curve between two tangent lines and a known chord length. It prompts for the chord length, the P.I. and points on the two tangent lines.

Prompts

Chord Length <100.00>: *press Enter*
[int on] Pick P.I. of curve: *pick a point*
[nea on] Pick Point on 1st Tangent Line: *pick a point*
[nea on] Pick Point on 2nd Tangent Line: *pick a point*

Pulldown Menu Location: Draw > Arc
Keyboard Command: 2tanlcl
Prerequisite: Tangent lines should be drawn before execution
File Name: \lsp\2tanlcl.lsp

3 Point

This command draws an arc between three points. The first point is the PC, the second is a point on the arc and the third is the PT. The points can either by picked on-screen or specified by point number.

Prompts

Pick PC point or point numbers: *101* (For point number 101.)
Pick Second point or point number: *102*
Pick PT point or point number: *103*

Pulldown Menu Location: Draw > Arc
Keyboard Command: 3PA
Prerequisite: None
File Name: \lsp\3ptarc.lsp

PC, PT, Radius Point

This command draws an arc between the PC point, radius point and PT point. The points can either by picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise. The program shows one direction and asks if it is correct. If you need the arc to go the other direction, enter No.

Prompts

Pick PC point or point number: *101*
Pick Radius point or point number: *102*
Pick PT point or point number: *103*
Is the direction of this arc correct ? No/<Yes>: *N*

Pulldown Menu Location: Draw > Arc
Keyboard Command: pca
Prerequisite: None

PC, Radius, Chord

This command draws an arc, given the PC point, radius length, chord length and chord bearing. The PC point can either by picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counter-clockwise. The program shows one direction and asks if it is correct. If you need the arc to go the other direction, enter No.

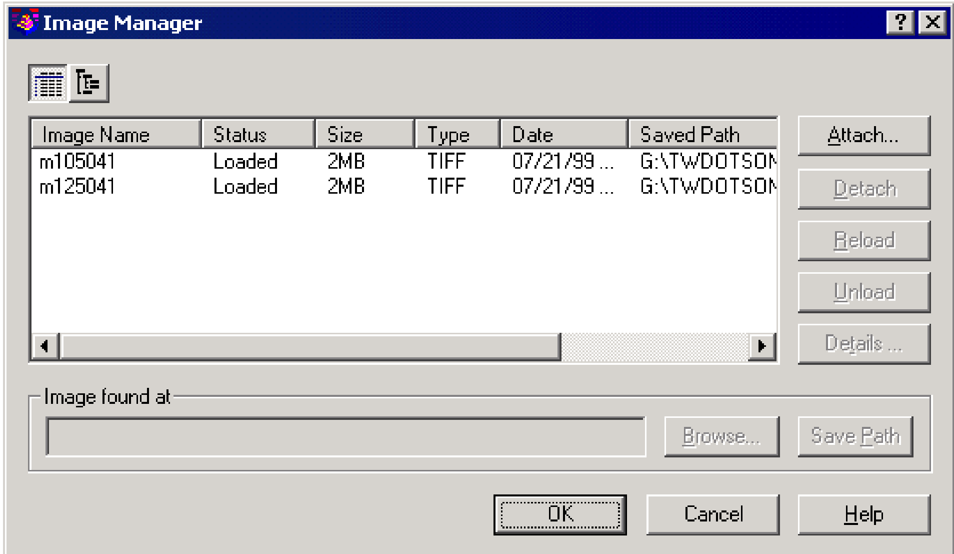
Prompts

Radius of Arc <-40.00>: *500*
PC Start Point ?
Pick point or point number: *pick a point*
Chord bearing or chord endpoint (<Bearing>/Point)? *Press Enter*
Enter Bearing (Qdd.mmss) <90.0000>: *145.1041* (for NE 45d10'41")
Chord Length <200.46>: 200
Is this arc in the correct direction (<Yes>/No)? *Press Enter*

Pulldown Menu Location: Draw > Arc
Keyboard Command: srcb
Prerequisite: None

Raster Image

This command allows you to manage raster images.



1 The Image Manager dialog box lists all the image files attached to the current drawing. You can view the parameters and details for selected images. You can attach new image files and detach, locate, reload, and unload existing images.

• List View: This button lists the image definitions attached to the drawing. Each image name appears only once regardless of how many times you attach (insert) the image. You can sort the list of images by name, status (loaded, unloaded, or not found), size, type (TIFF, for example), date, or the saved path and file name. By default, TakeOff displays the list alphabetically by image name.

To select multiple images, hold down SHIFT or CTRL while selecting items.

To sort the list alphabetically or numerically by a specific column, click that column's heading.

To change the width of the column, drag the line between the column headings to the right or left. The program saves and restores the settings when you reopen the dialog box.

To change an image name, select it and then click it again, or select it and then press F2. You cannot edit names of images that reside in external references (xrefs). Image names can include up to 255 characters and can contain letters, digits, spaces, and any special characters not used by Microsoft® Windows® or TakeOff. The image name can be identical to the file name, but changing the image name does not change the file name.

• Tree View: This button displays all the image definitions and the levels of nesting of images within xrefs. The top level of the tree view shows images that you attached directly to the drawing, images nested in block references, and the names of externally referenced drawings containing images. The names of the images attached to the externally referenced drawings appear nested within the drawing at the next tree level. To insert a copy of an already attached image, select it, and then choose Attach.

Tree view lists the image names only (not file names) and lists the image name just once, regardless of how many times you attach (insert) the image.

You can edit an image name by selecting it and then clicking it again, or by selecting it and then pressing F2. However, you cannot select more than one image at a time.

• Attach: This option displays the Select Image File dialog box. When you unload and then reload an image, the program draws that image on top. Images remain loaded or unloaded from one drawing session to the next.

• Detach: This option removes the selected image definitions from the drawing database and erases all the associated image objects from the drawing and from the display.

• Reload: This option loads the most recent version of an image or reloads an image that was previously unloaded. Reloading does not control whether the image is displayed, but it ensures display of the most current image.

• Unload: This option unloads image data from working memory without erasing the image objects from the drawing. It is recommended that you unload images no longer needed for editing to improve performance. An unloaded image cannot be displayed or plotted. You can selectively load and unload individual images from a working list of images associated with the drawing file.

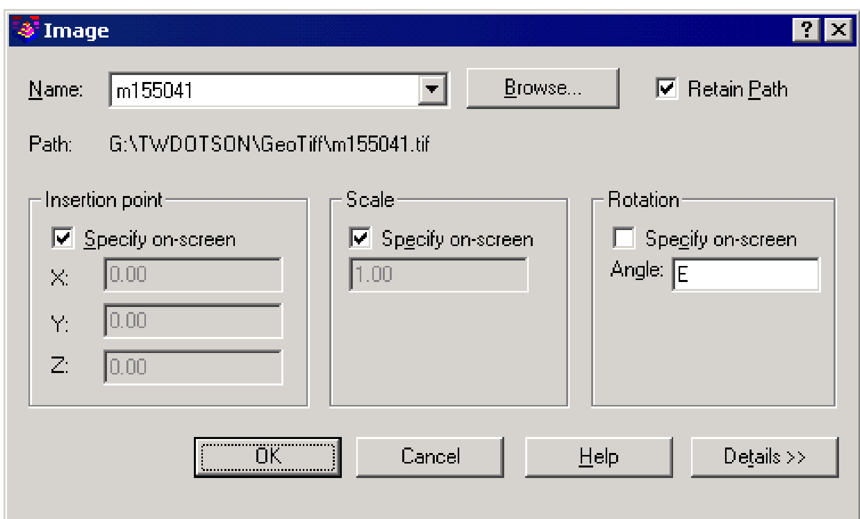
• Details: This option opens the Image File Details dialog box, which displays the image name, saved path, active path, file creation date and time, file size and type, color system, color depth, width and height in pixels, resolution, default size in units, and a preview image.

• Image Found At: This field shows the path of the selected image. If you select multiple images, this field remains blank. The path shown is the actual path where the image resides.

• Browse: This option opens the Select Image File dialog box (a standard file selection dialog box). The path you select appears under Image Found At.

• Save Path: This option stores the new path information. Press ESC while editing the path to restore the old path. If the program cannot find the referenced image in the new path, the image's status changes to Not Found. If you do not choose Save Path after editing the path, the program uses the original image path the next time you load the drawing.

2 Under the Image dialog box, you can attach an image.



3 In the Image dialog box, you must first identify the image and the path.

• Name: This field identifies the image you have selected to attach, either from the Select Image File dialog box (an unattached image) or from the list of previously attached images. To add another instance of an image file that is already attached, select the image name from the list and choose OK.

• Browse: This option opens the Select Image File dialog box (a standard file selection dialog box). If Show Preview is selected, the program displays a preview of the selected file.

• Retain Path: This option saves the path of the image file with the image definition. If Retain Path is not selected, only the image name is saved and TakeOff searches the Support File Search Path.

4 Under Insertion Point, you must specify the insertion point for the selected image. Specify On-Screen is the default. The default insertion point is 0,0.

- Specify On-Screen: This option directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter the insertion point in X, Y, and Z.
- X: This field sets the X coordinate value.
- Y: This field sets the Y coordinate value.
- Z: This field sets the Z coordinate value.

5 Under Scale, you must specify the scale factor of the selected image. Specify On-Screen directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter a value for the scale factor. The default scale factor is 1.

6 Under Rotation, you must specify the rotation angle of the selected image. If Specify On-Screen is selected, you may wait until you exit the dialog box to rotate the object with your pointing device or enter a rotation angle value on the command line. If Specify On-Screen is cleared, enter the rotation angle value in the dialog box. The default rotation angle is 0.

Prerequisite: Raster image

Keyboard Command: IMAGE

Place Image by World File

Function

This function allows you to insert Geo-Referenced TIF files into AutoCAD drawings. This process requires the presence of an accompanying TFW file. The TFW file contains information about the location and scaling of the actual raster image TIF file. This eliminates the guesswork in inserting, moving, and rotating raster images to the project area. You begin by selecting the TFW or JGW file to process. If the related TIF file is present in the same directory, the image will be inserted into the proper coordinates.

Prompts

Select World File: *choose existing .TFW or .JGW file*

Keyboard Command: geotiff
Prerequisite: None

Closed Polyline By Interior Text

This command allows you to create closed polylines from existing linework. Select all the entities (lines, arcs, or polylines) you would like to use, specify desired snap tolerance (for joining broken lines), then click inside the boundary you would like to close, and the command will generate corresponding closed polylines. Duplicate polylines are detected and are not created. The new polylines are always created on the current layer; the layers of the original linework are not used.

Prompts

1 Select polylines: **pick entitie**s

2 Select objects: **pick entities**

3 Select objects: **Press Enter**

4 Enter snap tolerance <0.0>: **enter a value**

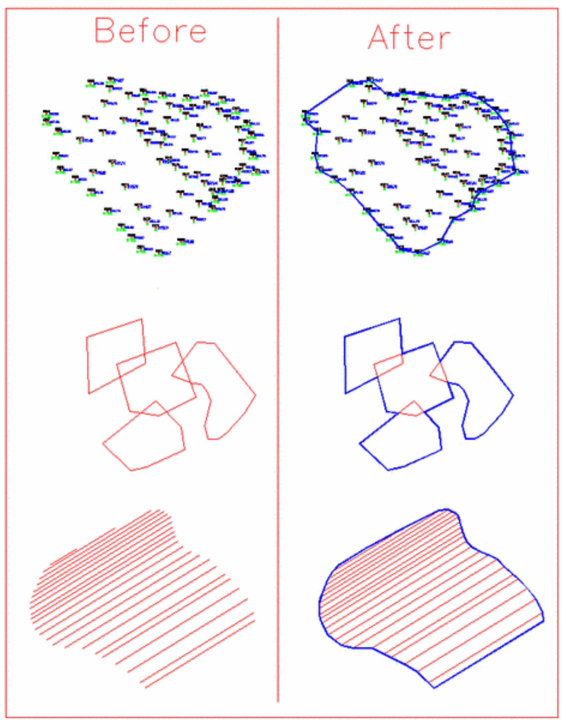
5 Pick an internal point: **pick a point**

Prerequisite: Entities on screen.

Keyboard Command: BOUNDPL

Shrink-Wrap Entities

This command creates a closed polyline which encloses a given set of entities. The resulting polyline is created in the current layer. The program works on either point entities or polylines. For points, the program creates a closed polyline through the points around the perimeter of the area defined by the points. For polylines, the shrink-wrap polyline follows the outside border of the selected polylines. The polylines that are processed have to be connected to be shrink-wrapped. The snap tolerance is the maximum gap that will be joined to make the closed polyline. For open polylines, as in the bottom figure, the Gap method works better, as it jumps across the gaps and connects the end points.



Prompts

Shrink-wrap across gaps or bounded linework only [<Gap>/Bound]? *G*
Shrink-wrap layer <FINAL>:
Select points and linework to shrink-wrap.
Select objects: *select entities to process*
Reading points... 46
Inserted 46 points.
Inserted 23 breakline segments
Perimeter reduction level 0-3 (0-None, 3-Most) <2>: *2*

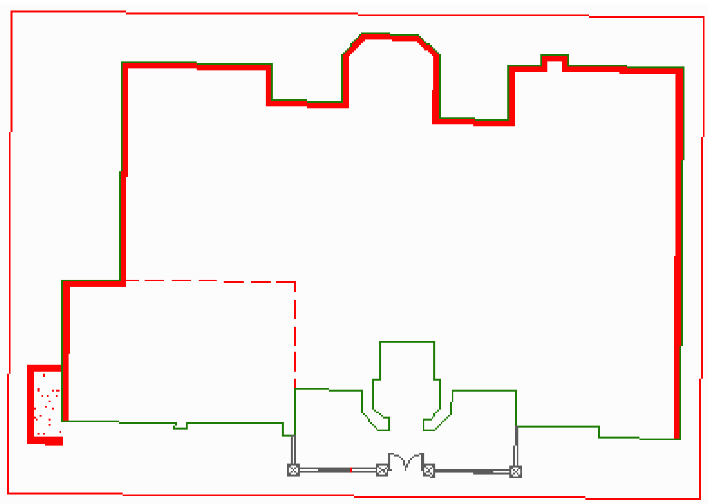
Reduce Perimeter Pass: 1 Removed: 5
Reduce Perimeter Pass: 2 Removed: 3
Reduce Perimeter Pass: 3 Removed: 4
Reduce Perimeter Pass: 4 Removed: 2
Reduce Perimeter Pass: 5 Removed: 1
Reduce Perimeter Pass: 6 Removed: 0
Create 2D or 3D Polyline [<2D>/3D]? *2D*

Pulldown Menu Location: Draw
Keyboard Command: swplines
Prerequisite: Entities
File Name: \lsp\scbpoly.arx

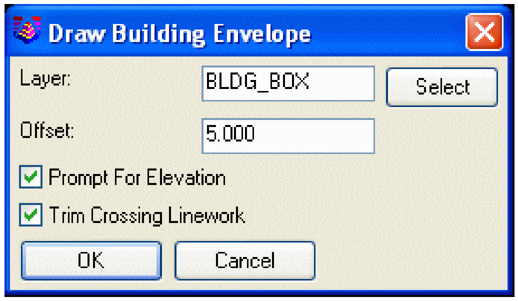
Building Envelope Polyline

Function

This command creates a rectangler polyline around selected linework. This can be used to give a building all one elevation.



Select the entities that make up the building. Next you will be prompted to name the layer and to set an offset distance (The above example is offset by 5 feet). Also, you can set the elevation of the envelope and trim crossing linework to ensure you have a flat pad.



Prompts

Draw Building Envelope dialog
Select building lines.
Select objects: *pick the linework that makes up the perimeter of the building*
Draw another building envelope [<Yes>/No]? *N*

Prerequisite: a pad

Keyboard Command: bldg_perim

Title Block

This function is different depending on your AutoCAD version

Function (AutoCAD R14)

This command draws a border and title block for the selected sheet size. The margins to useare specified at the bottom of the dialog. Margins are needed so that the border fits in theplotter's plotable area. For sheet 11x17 or smaller, a 1/2 inch margin is typical. For larger sheets, atypical margin is 3/4 inch. The *LIMITS* of the drawing can be set to the lower left and upper rightcorners of the border. To change the title block, edit the drawing TBLOCK.DWG in the \SUP directory.Click the toggle User Defined to set a custom sheet size. The default user defined size can be storedin the Configure TakOff command under the General Settings option. After the title blockis drawn, the contents can be edited using the Attribute Edit command under the Modify menu.The Change Scale button will change the scale for the title block and for the drawing.

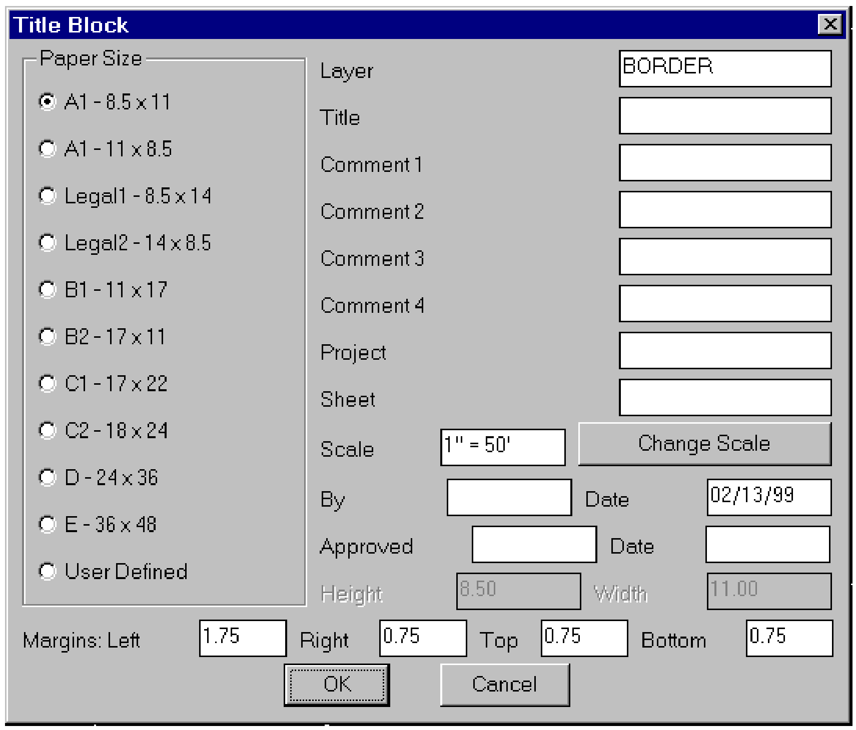
Function (AutoCAD R2000 and up)

This command draws a border and title block for the selected sheet size. At the top of thedialog, choose your horizontal scale and sheet size. The ∗other∗ choice at the bottom of each list willallow you to add your own scale or size if yours is not listed. Anything added to these lists will beretained for future use. Next, choose either "landscape" or "portrait" format. A blue rectangle next tothis choice shows you the difference. Below this, you can choose what layer to draw the borderand title block on. The margins to use are specified next at the bottom of the dialog. On the righthand side of the dialog. you can choose from several title blocks. As you choose each one, apreview will be shown below this list. This routine looks for all drawings named "tblock" in the\SUP directory. If you want to add your own title block, simply create a new drawing (or copy anexisting one) in the \SUP directory and give it a name that starts with tblock. Example: tblock22.dwgand tblock-Jones.dwg are both valid names for this routine, but "MyTitleblk.dwg" is not. Afteryou have made all your decisions in the dialog box, press OK. Depending on your

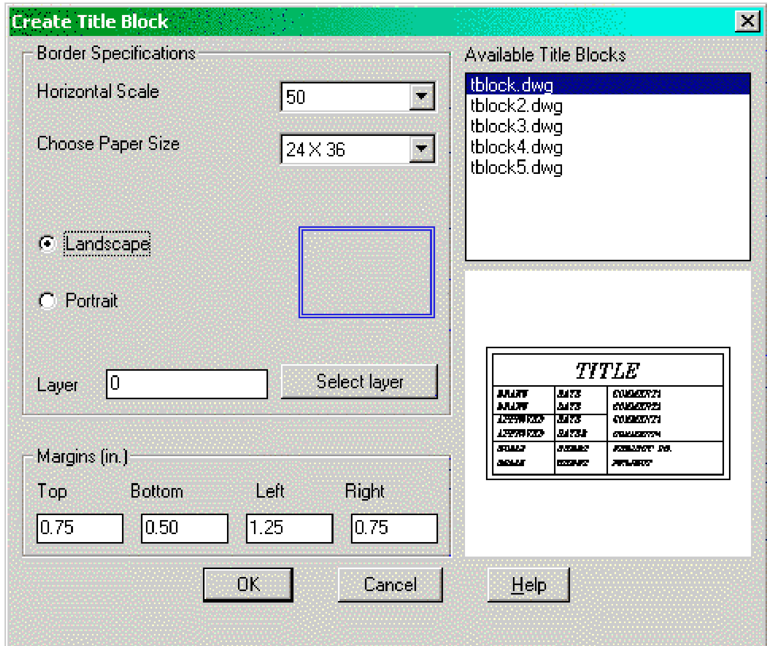
current zoomlevel, your drawing may be zoomed out to allow you to see the entire area that will be covered bythe drawing border. At this point, you have the border attached to your cursor and itis waiting for you to pick a point for insertion. As soon as you do this, a secondarydialog will appear for you to fill out theattributes associated with the particular title blockyou selected.

Prerequisite: set horizontal scale in Drawing Setup

<>**Keyboard Command:** tblock



Title Block Dialog for R14



Title Block Dialog for R2000 & up</>

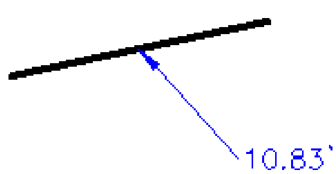
Distance with Leader

Function

This command labels the distance of a line or polyline segment at a point then draws a user specified leader line to point to the defining line. There is the ability for multi-segment leaders, and the option to align the label horizontal to the current view or parallel to the linework.

Prompts

Define distance by, Points/<Select line or polyline>: *select a line*
Pick point to start leader: *pick a point near the line*
Label Position: *pick a point*
Define distance by, Points/<select line or polyline>: *press Enter to end*



Keyboard Command: distlead
Prerequisite: None

Curve - Arrow

Curve - Arrow can be used to draw a section of contour line or create leader pointer lines. Curve - Arrow draws a Bezier curve through user specified points. After choosing endpoints, each time an intermediate points is picked the curve will be redrawn through all the points. There is an option to draw an arrowhead at the starting point. The arrowhead size is determined by the AutoCAD system variable "DIMASZ". In order to change this size, type DIMASZ at the AutoCAD command prompt. This routine also has a Zorro option which creates a Z leader curve.

Prompts

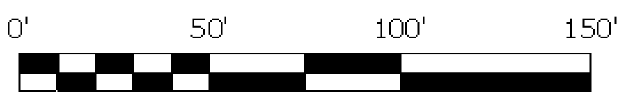
Create a Zorro (Yes/<No>)? *N*
Include an arrow (Yes/<No>)? *Y*
Enter the arrow head size <4.00>: *press Enter* This defaults to the DIMASZ system variable.
Pick a starting point: *pick a point*
Pick an ending point: *pick a point*
Pick an intermediate point (U to Undo): *pick a point*
Pick an intermediate point (U to Undo): *press Enter*

Examples of Curve - Arrow

Pulldown Menu Location: Draw
Keyboard Command: carrow
Prerequisite: None
File Names: \lsp\cir_num.lsp, \lsp\scadutil.arx

Barscale

This command draws a barscale. You will be prompted for the horizontal scale. The default value is set in the Drawing Setup command in the Settings menu.



Prompts

1 Insertion Point: **pick a point**

2 Horizontal scale <50.0>: **Press Enter**

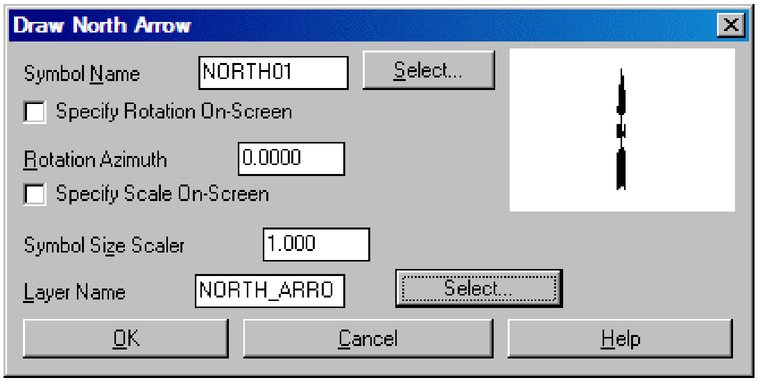
Prerequisite: None

Keyboard Command: BARSCALE

North Arrow

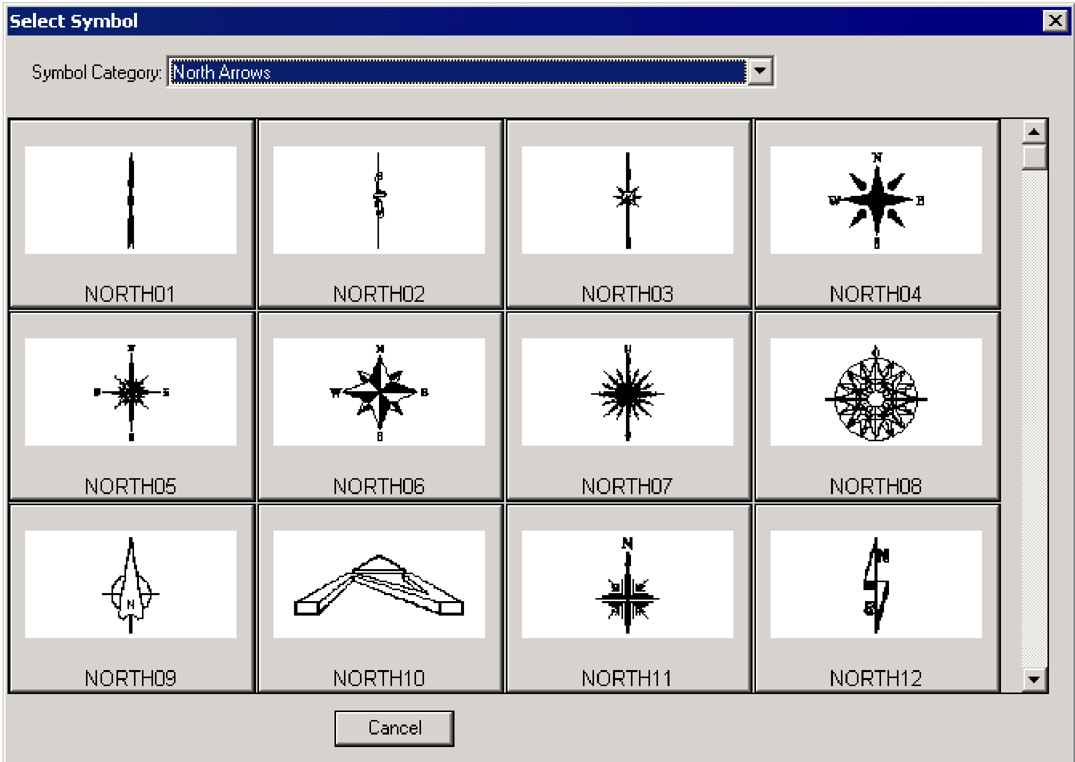
Function

This command inserts a north arrow symbol. You can select from several styles of arrows, and you can add your own by using the *Edit Symbol Library* command on the Settings menu.



Prompts

Draw North Arrow Dialog *choose an arrow symbol, layer and other variables*
Specify insertion point: *pick a point*



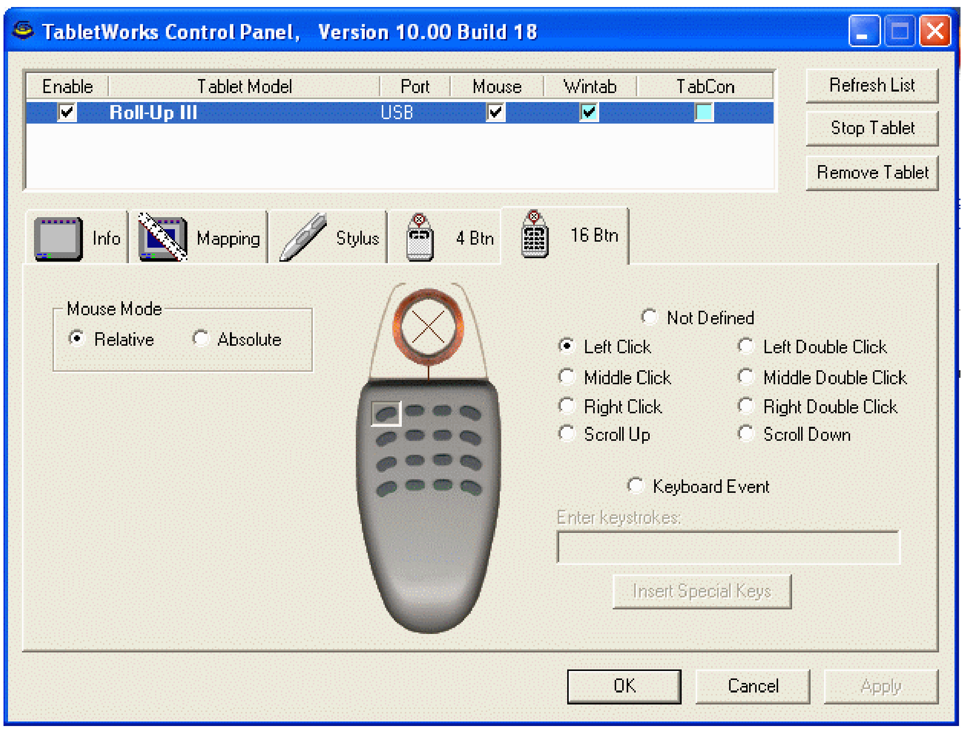
Keyboard Command: narrow
Prerequisite: None

Digitize Menu 6

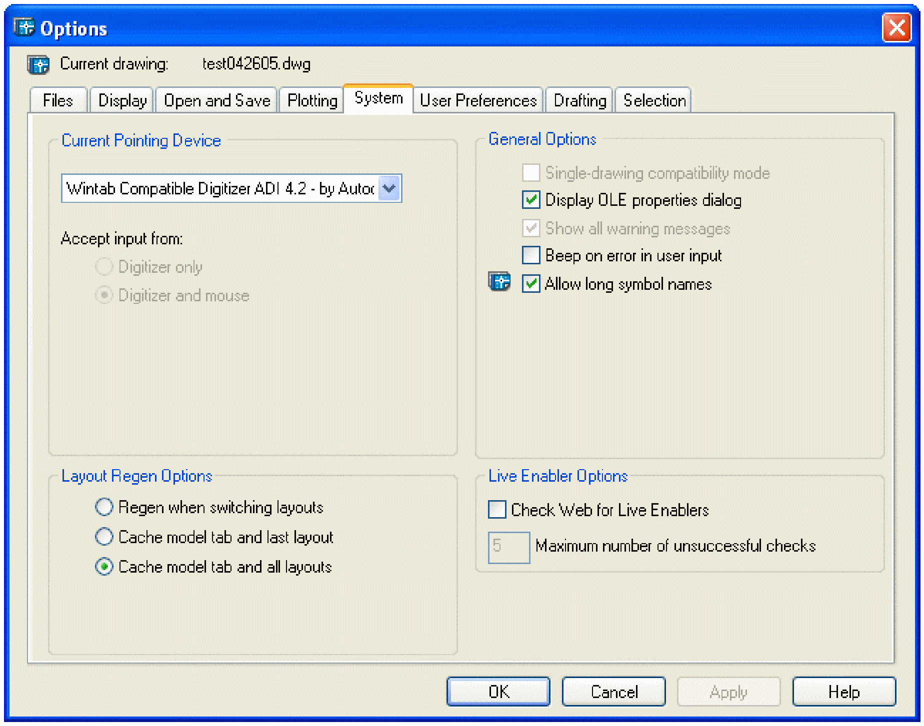
Digitizer Setup

Digitizing is the act of inputting data into the computer by tracing the data from a plan sheet. You need to have a digitizer board, puck, Carlson Takeoff, your computer and your plan to do digitizing. Wintab is a digitizer driver that lets you to use the digitizer cursor as both a digitizer cursor and a mouse. You need to install Wintab when you install Carlson Takeoff. Wintab can be downloaded from GTCO web site: http://www.gtcocalcomp.com/supportgtcosoftware.htm. Select the driver version that suits the type of your digitizer board well.

After you installed Wintab driver on your computer, you set up you digitizer to the correct point mode. In Windows 2000/XP, go to **Start->Settings->Control Panel->TabletWorks**, high light the **16-Btn Cursor**, and select **Mouse** as the **Pointing Mode**, which lets the digitizer cursor moves relatively to the screen coordinates. This step is indicated in the following **TabletWorks Control Panel** dialog.



The next is to set up the pointing device in Carlson Takeoff. Open up Takeoff and go to pull-down **Settings->Preferences**, click tab **System**, select **Wintab Compatible Digitizer** as **Current Pointing Device**, and set the **Accept input from** to **Digitize and mouse**. Please refer to the following **Options** dialog.



Now, you are ready to use your digitizer. On the bottom of the screen, there is a tray icon TABLET on the right side of MODEL. You can use accelerator key **F4** to toggle on/off the tablet.

Tablet On

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

Note: Function key [F4] can toggle on/off tablet.

Keyboard Command: tablet
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Tablet Off

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

Note: Function key [F4] can toggle on/off tablet.

Keyboard Command: tablet
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The

digitizer has been correctly set up.

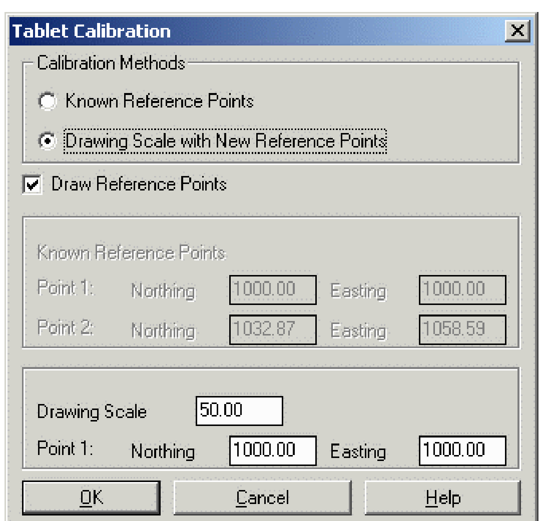
Tablet Calibrate

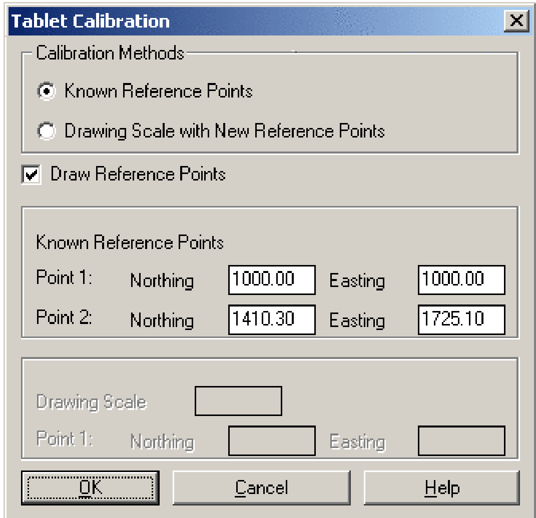
You can calibrate the tablet/digitizer in one of two ways: **Known Reference Points** or **Drawing Scale with New Reference Points**. Reference points are the foundations of whatever data you digitize into the computer. Takeoff bases everything from drawing location to drawing scale on the reference points you digitize.

Drawing Scale with New Reference Points method is very convenient when you don't know the precise coordinates of the entities on your drawing. As long as your can obtain the drawing scale from your plan, this method can establish a coordinate system relative to the position of the plan on the digitizer board. In addition to the drawing scale, you are required to enter a random coordinate for the first reference point, the default coordinate is (1000,1000). Takeoff would computer the coordinate of the second reference point that you pick based on the first point. The coordinates of these two reference points would be saved and will be display on the **Tablet Calibration Dialog** next time when you calibrate the tablet, so you can digitize back to the previous coordinates using **Know Reference Points** method if you are working on the same drawing, though you might have moved or rotated your drawing on the digitize board..

If you know the precise coordinates of two points, you can select **Known Reference Points** method, which establishes a coordinate system that is exactly match the coordinates in the field or on your drawing. Furthermore, Takeoff saves the coordinates of the two reference points from previous calibration and displays them on the **Tablet Calibration Dialog** next time when you calibrate the tablet. If you want to continue to work on the same drawing, you can use the **Know Reference Points** method with the saved coordinates to digitize back to your previous coordinates although you might have moved or rotated your drawing on the digitizer board.

For accurate takeoff calculations, choose two points that can be easily found in the field and are farther apart rather than closer together.





Prompts

Tablet Calibration Dialog
Specify the Calibration Methods. If you select Drawing Scale method, enter the drawing scale and the coordinate of the first reference point. Otherwise enter the exact coordinates of the first and second reference points.
Pick first reference point:*pick a point on the drawing*
Pick second reference point:*pick another point on the drawing*

Keyboard Command: digsetup
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Save Tablet Calibration

This command saves current tablet calibration to a file. You are prompted to enter a file name.

Keyboard Command: tablet1
Prerequisite:
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

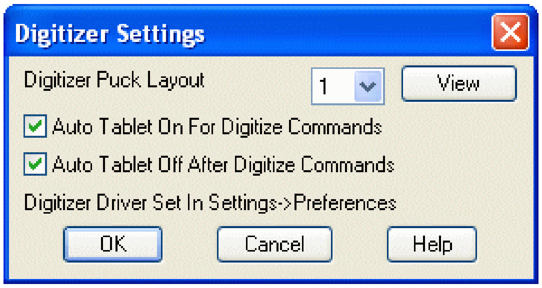
Load Tablet Calibration

This command restores the tablet calibration parameters from a file and load it into the current drawing. You are prompted to specify a file name.

Keyboard Command: tablet2
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. The calibration file should be associated to the current drawing, and the current drawing shouldn't have been moved on the digitizer board since last calibration.

Digitizer Settings

This command allows you to select the puck layout and set Auto On/Off features.



Auto Tablet On For Digitize Commands means after you select a digitize command your puck will automatically be put in Digitize Mode. If this is toggle off, then you will need to turn Tablet on separately from running a digitize command.

Auto Tablet Off After Digitize Commands means you will return to Mouse Mode after running a digitize command. Read below for more on Mouse and Digitize Mode.

Puck Layout

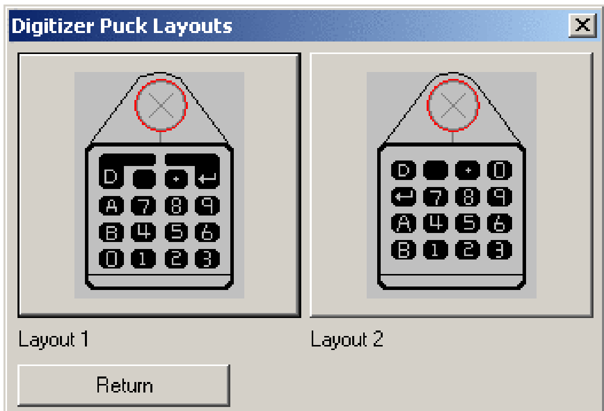
The 16-button puck can be used as either a mouse or a digitizer. It's very important to understand how the 16 buttons are mapped in both modes.

Mouse Mode:

When the tablet is off, the puck is in Mouse Mode. The top-left button is the left mouse click, and the top-right button is the right mouse click. The labels on the other buttons do not mean anything. All buttons are mapped as same as the buttons of the default pointing device in AutoCad . Please refer to AutoCad Reference manual for further information.

Digitize Mode:

When tablet has been calibrated and is on, the puck is in digitize mode. It is mapped as a small keyboard, which enables you to enter numerous values such as elevation, thickness and offset etc., and also provide you some functionality to digitize various entities. Currently there are two puck layouts in Takeoff, shown in the figure below. After you install Carlson Takeoff and finish setting up the digitizer, you go to the pull-down menu **Digitize->Puck Layout** to select a 16-button puck layout. A button mapping would be created and Takeoff would recognize the buttons as represented.



Layout 1 is Carlson Puck Layout, which is the most common layout used in Carlson Takeoff. Layout 2 is for users who don't have a Carlson Puck. If your puck is different than these two layouts, please contact Technical Support for help setting the mapping for your 16 button puck.

Prompts

Digitizer Settings Dialog
Specify the Digitizer Puck Layout to layout 1 or 2

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed.
Keyboard Command: dig_config

Digitize Existing

This sets the layer target to existing. Set this prior to running any digitizing command and anything you digitize will be assigned for your existing surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Keyboard Command: set_digit_exist

Prerequisite: none

Digitize Design

This sets the layer target to design. Set this prior to running any digitizing command and anything you digitize will be assigned for your design surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Prerequisite: none
Keyboard Command: set_digit_final

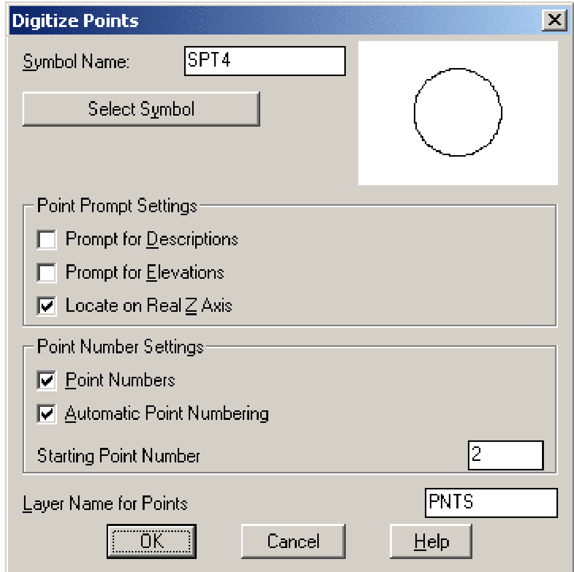
Digitize Other

This sets the layer target to other. Set this prior to running any digitizing command and anything you digitize will be assigned to the Other target. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Keyboard Command: set_digit_other
Prerequisite: none

Digitize Point

This command allows you to digitize individual points one at a time. The first time it prompts you the **Digitize Points Dialog** for entering point symbol styles, point prompt settings and number settings, starting point number and layer name. If you want to enter the elevation and description for each point, select **Prompt for Descriptions** and **Prompt for Elevations**. After having digitized a point, you can continue to digitize next point by picking the point on the drawing. The command defaults to the last layer name, point symbol, elevation, description and the last point number plus 1. If you have finished digitizing points, press **Enter** to finish.



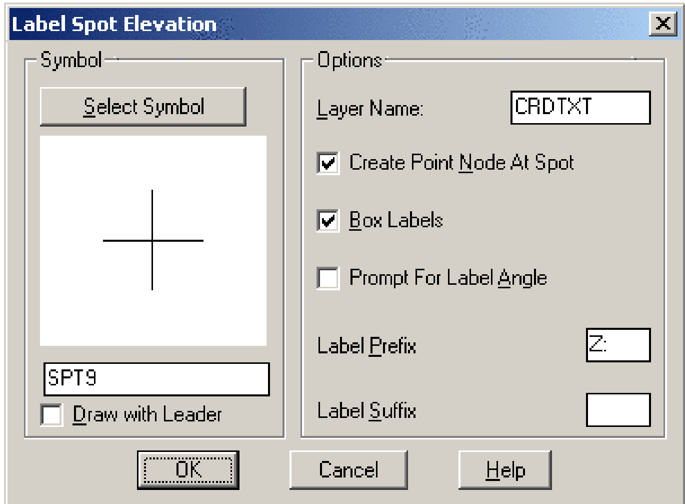
Prompts

Digitize Points Dialog
Specify a layer name and select the point symbol, point prompt settings and number settings.
Pick point to create (Enter to end): *pick a point on the drawing*
Select/<Enter Point Elevation <>>**:***enter the elevation or type <Select> to select the elevation text on the screen*
Enter Point Description <>**:***enter the point description*
Result like "N: 1231.16 E: 1099.17 Z: 30.00" would be display on the command line, and a point would be drawn on the screen with the text of its number, elevation and description.
Pick point to create (Enter to end):*pick next point or press Enter to finish digitizing points*

Keyboard Command: dig_pt
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize Spot Elevation

This command allows you to label points with their elevation. The point can either be digitized from a drawing, picked on a screen or specified by a point number. The command first prompts you the **Label Spot Elevation Dialog** for entering layer name, label prefix and suffix and symbol types etc. Click OK to start. After specifying the point, the command prompts you to enter the elevation if its elevation is unknown and then pick an angle from the location of the point to label the elevation. You can repeat labeling points until you press **Enter** to finish.



Prompts

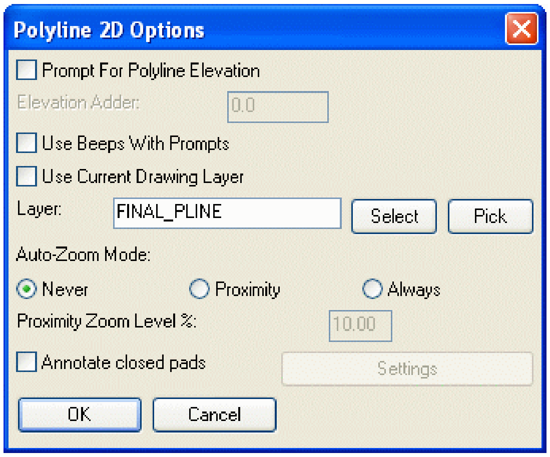
Label Spot Elevation Dialog
Specify a layer name, label prefix and suffix and select the spot symbol.
Point to Label ?
Pick point or point number: 2 *(enter a point number)*
PointNo. Northing(Y) Easting(X) Elev(Z) Description
2 1231.16 1099.17 30.00 bb
Note: if the point number you entered is not in the drawing, you will be prompted again to pick point or enter a point number.
Elevation <30.000>: *press enter*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *pick a point on the drawing*
Elevation <0.000>: *enter elevation*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *press enter to finish*

Keyboard Command: labspot
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize 2D Polyline

A 2D polyline is a line of connected points that have the same elevation. This command lets you digitize a 2D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 2D Options Dialog** for entering the layer name. **Prompt For Polyline Elevation** option allows you to enter the elevation for each polyline,

otherwise all 2D polylines have 0.0 elevation. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points. There are three ways to enter a layer name, **Use current drawing layer**, **Select** from a list of layer name, or **Pick** an entity on the screen to get its layer name. While digitizing a polyline, the command keeps prompting you to pick next point until your **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself . If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts your to digitize another polyline until you **press B** or **enter No**.



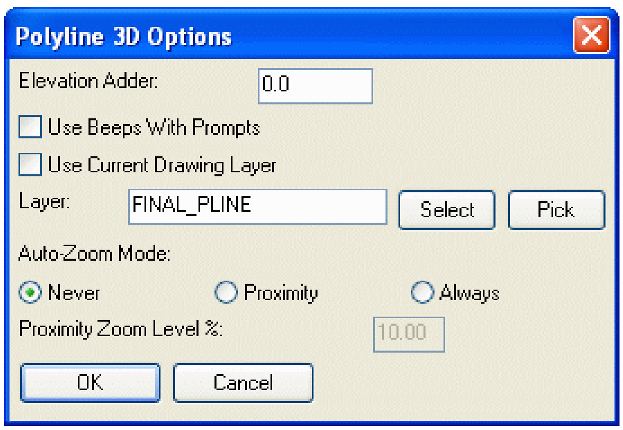
Prompts

Polyline 2D Options Dialog
Enter the layer name and select the options of Prompt For Polyline Elevation and Auto-Zoom mode etc.
Enter default elevation <0.00>: *100*
First point: *pick a point on the drawing using puck*
Segment length: 0.00, Total length: 0.00
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 119.03, Total length: 119.03
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 121.76, Total length: 240.80
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 115.23, Total length: 356.03
Close[A]/Undo[B]/Pick next point (Enter to end): *press enter to finish digitizing or press A to close the polyline*
Digitize Another FINAL Polyline [Yes(A)/<No(B)>]? *press A on the puck or enter Yes on the keyboard to digitize next 2D polyline, press B on the puck or enter No on the keyboard to finish digitizing 2D polyline.*

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.
Keyboard Command: dig_2dp

Digitize 3D Polyline

A 3D polyline is a line of connected points that have various elevations, and the slope between points is constant. It can be used in defining pads, excavations, drainage ditched and slopes from proposed design features to meet existing site conditions. This command lets you digitize a 3D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 3D Options Dialog** for entering the layer name. **Elevation Adder** allows you to truncate the elevations you have to enter in by add a given amount to them. There are five ways to enter elevations: known elevation of the point, interpolate, slope from previous point, ratio from previous point and degree from previous point. You can choose one of the methods between picking points. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points. While digitizing a polyline, press A to interpolate the elevation or B to enter it in. The command keeps prompting you to pick the next point until your **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself. You can also use AutoCAD's OSNAP command to pick points by pressing the decimal [.] button on the digitizer puck. If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts your to digitize another polyline until you **press B** or **enter No**.



Prompts

First point:
Interpolate[A]/screen Pick/<Elevation[B]> <0.00>: 256

Z: 256.00
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): Pick point
Slope/Ratio/Interpolate[A]/Degree/screen Pick/<Elevation[B]> <256.00>: A

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: Pick point
This point elevation will be interpolated upon completion.
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: 279

Z: 279.00, Hz dist: 30.01, Slope dist: 37.81, Slope: 76.6% Ratio: 1.3:1

Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): Pick point
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: Press Enter

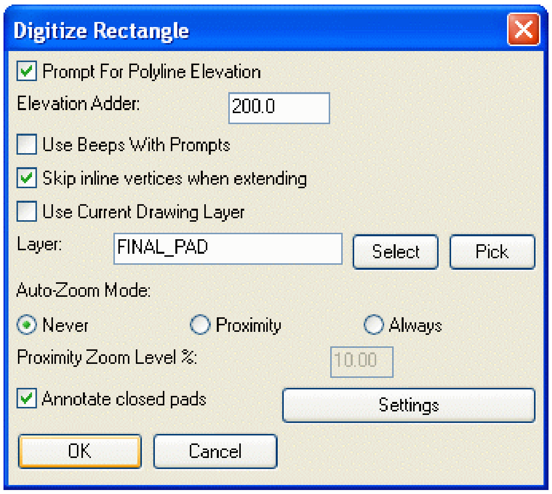
Z: 279.00, Hz dist: 24.18, Slope dist: 24.18, Slope: 0.0% Ratio: 0.0:1
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): A

Digitize Another EXIST_PLINE Polyline [Yes(A)/<No(B)>]? B
<Tablet Off>

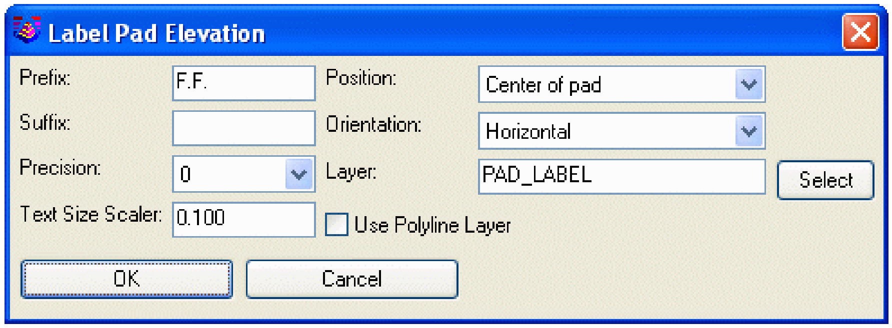
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.
Keyboard Command: dig_3dp

Digitize Rectangle

This command enables you to quickly create rectangles while digitizing. In the dialog, you can pick to set elevations to the rectangles, otherwise all rectangles will have 0.0 elevation. The Elevation Adder will be added to the value you enter in for the prompt "Enter polyline elevation <0.00>:". For example, if you know all the rectangles you are creating are in the 200s for elevation, you can put in this value for the Elevation Adder and simply put 46, 54, 57, etc. when prompted, and your rectangles will end up with the elevations of 246, 254, 257 etc. There are three ways to enter a layer name, Use current drawing layer, Select from a list of layer name, or Pick an entity on the screen to get its layer name. Auto-Zoom mode would automatically zoom the display to center around the last point when you get near the edge of the screen while picking points.



Annotate closed pads will label your rectangles according to the Settings button/dialog shown below:



In this dialog, you can enter in a Prefix or a Suffix to the elevation, and determine the labels position, orientation, precision out to 5 decimal places, its layer, and text size.

Prompts

Command:
DIG_RECT
Target surface: Design

Digitize Rectangle Dialog
Make any chances you desire in the above dialogs.

Enter polyline elevation <0.00>: *200*
First point: *pick a point on the drawing using puck*
Segment length: 0.00, Total length: 0.00
Close[A]/Undo[B]/Osnap[.]/Pick next point: *pick next point*
Segment length: 1105.96, Total length: 1105.96
Close[A]/Undo[B]/Osnap[.]/Pick next point: *pick next point*
Segment length: 426.83, Total length: 1532.79, Area: 236021.59
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *After 3 points you can press (A) for Close to create a rectangle*
Digitize Another FINAL_PAD Polyline [Yes(A)/<No(B)>]? *B for No*

Prerequisite: a digitizer
Keyboard Command: DIG_RECT

Digitize Perimeter

Perimeter is a 2D polyline that all points on it have the same elevation. It can be used as boundary polyline of your targets on your drawing. This command allows you to digitize a perimeter by picking points on the drawing. While digitizing a polyline, the command keeps prompting you to pick next point until your **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself . If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a perimeter, the command prompts your to digitize another polyline until you **press B** or **enter No**.

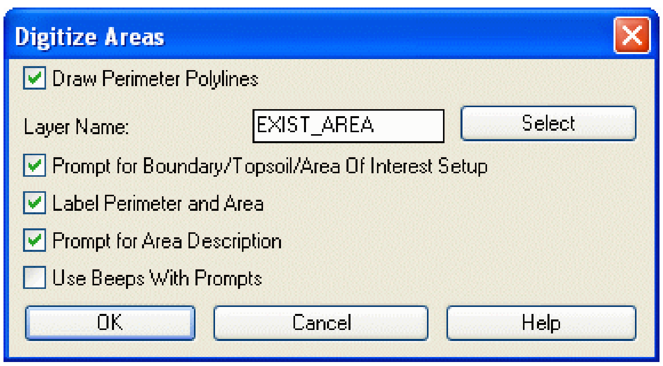
Prompts

First point: *pick a point on the drawing using puck*
Segment length: 0.00, Total length: 0.00
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 104.27, Total length: 104.27
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 153.14, Total length: 257.41
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 104.89, Total length: 362.30
Close[A]/Undo[B]/Pick next point (Enter to end): *press Enter to finish the perimeter, or press A to close the perimeter*
Digitize Another PERIMETER Polyline [Yes(A)/<No(B)>]? *press A or enter Yes to continue digitizing another perimeter, press B or enter No to finish digitizing perimeters.*

Keyboard Command: dig_perim
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize Areas

This command allows you to find an area in digitize mode. With the puck, pick around the area you wish to calculate. If Draw Perimeter Polyline is toggled on then the linework of your perimeter will be displayed. You can then set the Layer Name and choose to label the Perimeter and Area and enter in an Area Description. You can also set the area you created as a Boundary, Topsoil, or Area of Interest.



Prerequisite: a digitizer
Keyboard Command: dig_area

Digitize Contour Polyline

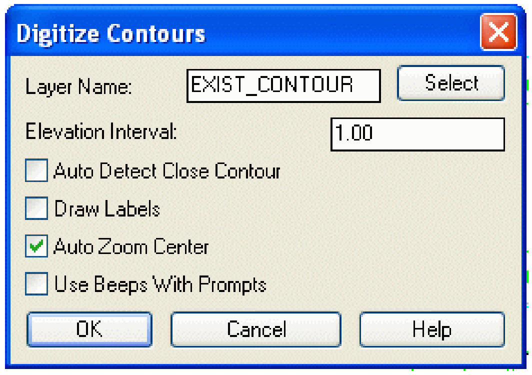
A contour is a line of points with a constant elevation, representing the natural contour of the site. In Takeoff, there are two layer targets: Existing Ground Surface and Design Surface. Contour Polyline has two sub-command to digitize contour lines into Existing Contour and Final Contour layers directly for assigning them easily into Existing Ground Surface and Design Surface in the future analysis.

There are two ways to digitize contour lines: sketch mode or point mode. You can start digitizing a contour with one mode and switch to the other during digitizing the contour. Sketch mode uses more points than pick mode. In general, we recommend using pick mode to digitize the straight parts of lines because it reduces the number of points and speeds up Takeoff's calculations, but using sketch mode to digitize the curved parts because it is fast and accurate.

Function

This command lets you digitize contours as polylines one at a time. The first time it prompts you the Digitize Contours Dialog. Enter the layer name or select it from a list of existing layer. Look at your plans and determine an elevation interval that is between most of the contours and enter it in the Elevation Interval field. You are able to modify both the value and the direction of the elevation interval between digitizing contour lines, using the buttons on the puck. To have Takeoff automatically close contours whose beginning and ending points are within a specified range, check the Auto Detect Close Contour. Draw Labels would draw the elevation at the starting point of the contour. In Pick mode, if you want the Takeoff to automatically zoom the display to center around the last point when you get near the edge of the screen while picking points, check the Auto Zoom Center. Click OK to start digitizing.

If this is your first time digitizing a contour, you are defaulted to the Pick Mode digitizing, otherwise you would be defaulted to the last digitize mode. If you want to use the other digitize mode, press 0 on the puck or enter 0 from the keyboard. Place your cursor at one end of the contour line and begin digitizing the line. While digitizing a line, you can force a contour to close on itself by pressing A on the puck to end the contour and connect the last point to the first point, remove a mistake by pressing B on the puck, or switch to the other digitize mode by pressing 0. During Sketch Mode digitizing, you can stop digitizing by pressing Pick or Enter button on the puck, take some rest or changes, and start sketching again. At the end of the contour line, **press Enter** on your puck or keyboard. The contour is completed, and the elevation for the next contour is automatically incremented. You would be asked to digitize next contour. If you press A on the puck or enter Yes on the keyboard, you can digitize another contour, or press B on the puck or enter No on the keyboard to finish digitizing contours.



Prompts

Digitize Contours Dialog
Enter Layer Name, Elevation Interval, and toggle on/off Auto Detect Close Contour etc.
Increment(1.00)[A]/Direction(+)[B]/Elevation <573.00>: 450 *(enter elevation or press Enter to accept current value)*

Start Digitizing...

Sketch[0]/Pick the first point: *pick a point to start Pick Mode digitizing (press 0 to switch to Sketch Mode)*
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): 0(press 0 on the puck or enter 0 on the keyboard to use Sketch Mode)

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag
Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)
Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)
Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag again
Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): 0 *(press 0 on the puck or enter 0 on the keyboard to use Pick Mode)*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): press Enter to finish digitizing

Digitize Another Contour [<Yes(A)>/No(B)]? B(press B to finish digitizing)

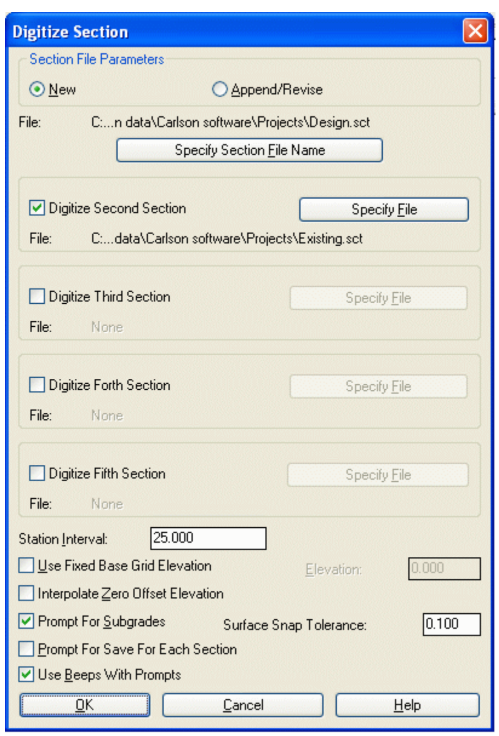
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: digcont_exist, digcont_final

Digitize Sections

This command allows you to digitize section lines and store the section data in the section file you have specified. The command first prompts you the **Digitize Section Dialog**. Enter the section file name and determine if you want to digitize second and third sections at the same station. Look at your plans and determines the station interval, which is used to automatically default to the next station value when digitizing a series of stations. If the grids at all the stations have the same base elevation, toggle on Use Fixed Base Grid Elevation. You can also toggle on Interpolate Zero Offset Elevation, Prompt for Subgrades, Prompt for Save for Each Section and Use Beeps with Prompts. Surface Snap Tolerance sets the maximum distance that the program will automatically snap the tie back point between the subgrade and design surface. Click OK to start digitizing.

Takeoff prompts you to calibrate the section sheet before you digitize the section lines. You pick three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Place your cursor at one end of the section line and begin digitizing the line. While digitizing a line, you can remove a mistake by **pressing A** on the puck or **entering Undo** on the keyboard. At the end of the section line, **press Enter** on your puck or keyboard. The station is completed, and the station value is automatically incremented. The command would prompts to digitize next section. You can **press A** on the puck or **enter Exit** on the keyboard to finish digitizing. If you want to continue to digitize next section, **press Enter** or enter the new station number. For every station after the first one, you can calibrate the grid sheet by picking one reference point and specify its offset and elevation. After you digitize the section lines on your drawing, all the section data would be saved in a section file (.sct).



Prompts

Digitize Section Dialog
Enter Section File Name, Station Interval, and toggle on/off Use Fixed Base Grid Elevation etc.
Section station to digitize <0.000>: *press Enter to start with station 0.0 or enter a station number*
Calibrate section sheet
Pick First section sheet reference point: *pick a grid point of this station on your drawing*
Enter offset <0.0>: *press Enter to accept the offset or enter the offset of the point to the centerline*
Enter elevation: 1030 *(enter the Elevation of the reference point)*
Pick Second section reference point: *pick the second grid point*
Enter offset: 0 *(enter the offset of the point to the centerline)*
Enter elevation: 1040 *(enter the Elevation of the reference point)*
Pick Third section reference point: *pick the third grid point*
Enter offset: 50 *(enter the offset of the point to the centerline)*
Enter elevation: 1040 *(enter the Elevation of the reference point)*

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible
RMS Error: 11.49
Standard deviation: 2.38
Largest residual: 14.08
At point: 2
Second-largest residual: 14.08
At point: 1

Digitize break point for DRAWING1 section 0.000 (Enter to end): *pick a point on the section line*
Offset: -39.81 Elev: 1028.80
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: -9.94 Elev: 1030.03
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: 49.44 Elev: 1034.93
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *press Enter to finish*
Save changes to DRAWING1 section 0.000 [<Yes(A)>/No(B)]? A *(press A or B)*

Exit[A]/Section station to digitize <50.000>: 200 *(enter next station number)*

Calibrate next section
Pick section reference point: *pick a grid point of the station on your drawing*
Enter offset <0.00>: *press Enter to accept the offset or enter the offset of the point to the centerline*
Enter elevation <1030.00>: 1020 *(enter the Elevation of the reference point)*
Digitize break point for DRAWING1 section 200.000 (Enter to end): *pick a point on the section line*
Offset: -40.40 Elev: 1008.07

Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: -5.38 Elev: 1019.98
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: 27.86 Elev: 1030.02
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: 50.33 Elev: 1035.80
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *press Enter to finish*
Save changes to DRAWING1 section 200.000 [<Yes(A)>/No(B)]? A *(press A or B)*

Exit[A]/Section station to digitize <250.000>: A *(press A to finish or enter the station number to continue)*

Keyboard Command: digxsec
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Digitize End Areas

There are two types of end areas: cut area and fill area. This command allows you to digitize both cut area and fill area on the drawing and writes data to a .ew file. The command first prompts you to calibrate the section sheet by picking three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals .Corners on the section grid are preferred reference points. Then it prompts you to digitize the cut area and fill area respectively. Place your cursor at one end of the end area and begin digitizing the outline of the area. At the end of the section line, **press Enter** on your puck or keyboard. The end area is completed, and its area is printed on the command line, and you are prompted to digitize next end area. After you finish all the end area at one station, accumulated cut area and fill area are computed and printed out on the screen. All data of cut area and fill area at every station would be saved in the area file (.ew) that you have specified.

Prompts

Calibrate section sheet
Pick First section sheet reference point: *pick a point on the drawing*
Enter offset <0.0>: *press Enter to accept the offset (or enter the offset of the point to the centerline)*
Enter elevation: 1020 *(enter the Elevation of the reference point)*
Pick Second section reference point: *pick a point*
Enter offset: 0 *(enter the offset of the point to the centerline)*
Enter elevation: 1030 *(enter the Elevation of the reference point)*
Pick Third section reference point: *pick a point*
Enter offset: 50 *(enter the offset of the point to the centerline)*
Enter elevation: 1030 *(enter the Elevation of the reference point)*

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible

RMS Error: 11.69
Standard deviation: 2.40
Largest residual: 14.29
At point: 2
Second-largest residual: 14.29
At point: 3

Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area*, 0∗(0.211129 1030.76)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area*, 1∗(11.5804 1030.49)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area*, 2∗(17.8643 1030.73)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area*, 3∗(19.0216 1032.35)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area*, 4∗(-0.777246 1030.75)
Digitize cut area (Enter to end): *press Enter to finish*
End area: 17.2312
Accumulated Cut Area: 17.2312
More Cut Areas [Yes(A)/<No>(B)]? *press A to digitize more Cut Areas, or press B to finish digitizing Cut Areas.*

Accumulated Cut Area: 17.2312
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area*, 0∗(-18.9614 1029.65)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area*, 1∗(-18.1315 1030.75)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area*, 2∗(-11.9592 1030.49)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area*, 3∗(-2.06761 1030.72)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area*, 4∗(-10.0082 1030.01)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area*, 5∗(-18.531 1029.67)
Digitize fill area (Enter to end): *press enter to finish*
End area: 8.64646
Accumulated Cut Area: 8.64646
More Fill Areas [Yes(A)/<No>(B)]? *press A to digitize more Fill Areas, or press B to finish digitizing Fill Areas.*
Accumulated Cut Area: 8.64646
Total Cut Area: 17.2312
Total Fill Area: 8.64646
Store data to file [<Yes>(A)/No(B)]? *press A or B*

Opened file: C:\Program Files\Carlson TakeOff 2004\DATA\Drawing1.ew
Station Number: 1 *(enter Station Number)*

Data Stored in file: C:\Program Files\Carlson TakeOff 2004\DATA\Drawing1.ew
Digitize another station [<Yes>(A)/No(B)]? B (*press A or B)*

Prerequisite:
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.<>
Keyboard Command: digendar

Elevate Menu 7

Change Elevations

This command will change the elevation of selected Entities. It can move the entity to a specified elevation from it's current elevation (absolute) or do a differential change by adding or subtracting a value from it's current elevation. If Carlson TakeOff points are selected, their attribute text and z axis coordinate are changed.

Prompts

Ignore zero elevations (<Yes>/No)? *Press Enter.* If you answer *No*, then entities with elevation 0 will be changed.
[A]bsolute or [D]ifferential Change <A>: *A*
Select/<Enter Elevation <0.0000>>: *125*
Change Layer for changed entities [Yes/<No>]: No
Elevation to change to:

By using the Absolute option all entities selected are changed to the elevation 125.

Select Entities for elevation change.
Select objects: *C*
First corner: *(pick point)*
Other corner: *(pick point)*
Select objects: *[Enter]*

Keyboard Command: chgelev
Prerequisite: Something to change

Set Polyline to Elevation

This command allows you to assign elevations to one or more polylines. The elevation can be assigned by entering in the value or by picking a text entity that has the elevation.

Prompts

Select/<Enter Elevation <0.0000>>: Select a text entity or type in an elevation. Press enter for the default elevation in brackets.

Select Polyline for elevation change. Pick on the screen a polyline you wish to change such as:
LWPOLYLINE
Done.
Set another polyline [<Yes>/No]? Press Y to pick another polyline to assign an elevation to. Type in N to finish the command.

Keyboard Command: set_pline_z
Prerequisite: A polyline and an elevation to assign it.

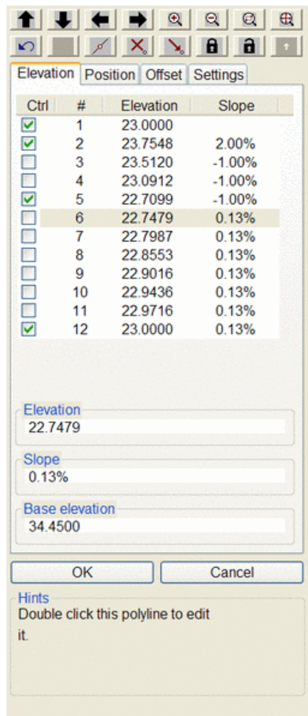
Edit-Assign Polyline Elevations

This command allows very precise control of 3D polylines, specifically in the ability to edit vertex elevations, as well as add, delete, or move vertices. You can also control the location of polyline vertices as defined by the station

and offset of the vertices relative to a Centerline.

Polyline vertices are designated as either Control or Free vertices. The elevation of Control vertices are set and held, the elevations of Free vertices are interpolated. In the drawing, control vertices are shown with red boxes, along with their vertex number and elevation. Free vertices are displayed with blue boxes and are not annotated.

When you run the command, you are first prompted to select a polyline to edit. When you pick a polyline to work with, the following control panel appears on the left side of your screen.



The top row of buttons across the top of the control panel are used to manipulate the view in the drawing with various Zooming and Panning options. The second row of buttons will change as you select different tabs, but are essentially used to add vertices, delete vertices, or pick elevations or locations for vertices.

The four tabs in the panel provide access to control of polyline vertex Elevation, Position, Offset and Settings.

Elevation: This tab displays the vertices of the polyline, each with a check box to set whether it is a control vertex or free, its assigned number, its elevation, and the slope from the previous vertex to that vertex. Selecting a vertex highlights its grip in the drawing. Once selected, you can enter an elevation or slope for that vertex in the spaces below the list, thereby automatically setting the vertex to a control vertex. The Base Elevation is used to adjust the elevations of all the vertices simultaneously.

Position: The Position tab displays the coordinates of each vertex. To move a vertex, you can type in new coordinates, use the Pick Position icon to specify a new location for the vertex on the screen, or you can grip the vertex and drag it to a new location.

Offset: The Offset tab requires the selection of a Centerline to reference. Once a Centerline is designated, the

Station, Slope, and Offset of each vertex relative to the Centerline is displayed and can be edited.

Settings: The Settings tab provides control over various overall options pertaining to the use of the command. For example, hiding free vertices and setting how to report your slopes between vertices.

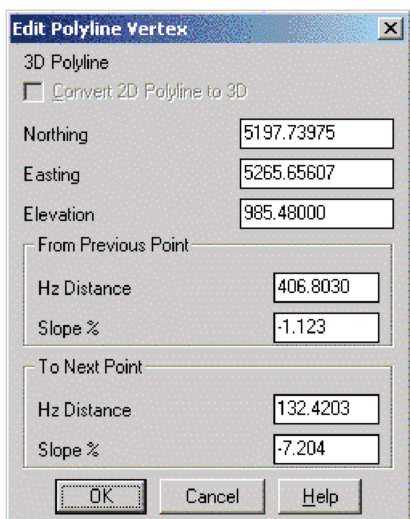
Right-click menu: There is a right-click menu available at all times which also gives access to a variety of functions and settings.

Keyboard Command: edit_pline_z
Prerequisite: Polylines with vertexes

Edit Polyline Vertex

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution you will be asked to select a polyline to edit. Upon selection a temporary marker will be placed at all of the vertices of the polyline, making them easy to distinguish. Then pick near the vertex you wish to edit, and the following dialog appears.

At the top of the dialog it identifies the type of polyline, being 2D or 3D. In the case of 2D polylines it allows you convert the polyline. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example if you change the horizontal distances, the coordinates will change.



Prompts

Select polyline vertex to edit: *pick a polyline at the point to be modified*
Pick or enter position <5264.23,5048.21>: *pick a point*

Enter elevation <0.00>: *Press Enter*
Select polyline vertex to edit: *Press Enter to end*

Keyboard Command: editpl
Prerequisite: A polyline.

2D to 3D By Surface Model

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge breakline could be generated from contour lines by drawing a 2D polyline along the ridge and across the contours. Then this command could grab the contour line elevations along the polyline to make a ridge breakline.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in triangulation (.flt or .tin) file. If you use a file, then you can also use the polyline's current elevation as a vertical offset from surface.

Prompts

By Screen Entities:
Source of surface model [File/<Screen>]? *Type S for Screen*
Select polylines to convert.
Select objects: *select the polyline(s) to convert*
Select surface 3DFaces, lines and polylines.
Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Reading points ... 692

Keep existing polylines [Yes/<No>]? *Press Enter*
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.
Set layer name for converted polylines [Yes/<No>]? *Press Enter*
This allows you to assign the new polyline to a layer.
Converting polylines ...
Converted 1 polylines.

By a .flt or .tin File:
Source of surface model [<File>/Screen]? *Type F for File*
Select polylines to convert.
Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Use current polyline elevations as vertical offset from surface [Yes/<No>]? *Press Enter*
This will offset the new polyline by its current elevation. That is, if a polyline has an elevation of -4 and the surface you are converting it to has an elevation of 800, then saying Yes will drape the polyline at an elevation of 796.
Keep existing polylines [Yes/<No>]? *Press Enter*
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.
Set layer name for converted polylines [Yes/<No>]? *Press Enter*
This allows you to assign the new polyline to a layer.

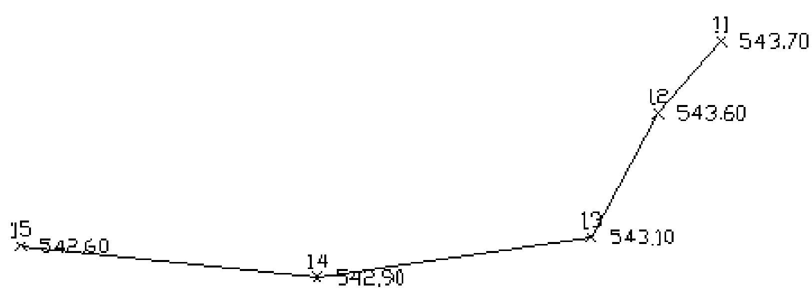
Keyboard Command: 2dto3dp
Prerequisite: A polyline and surface lines or grid file or triangulation file.

2D to 3D Polyline by Points

This command converts a 2D polyline into a 3D polyline by using the elevations of points. At each vertex of the polylines, the program looks for a point with elevation at the same x,y location. The points can be Carlson point blocks or AutoCAD POINT entities. This routine can be useful if the linework is created in 2D at zero elevation, and points with elevation are located along the linework. Then the linework can be converted into 3D polylines with this command. For example, a centerline polyline with arcs may need to be created in 2D for stationing because AutoCAD does not allow arcs on 3D polylines. To use this polyline as a breakline in surface modeling, this command can convert the polyline into a 3D polyline.

Prompts

Select points and polylines.
Select objects: select polylines to convert and the points with elevation

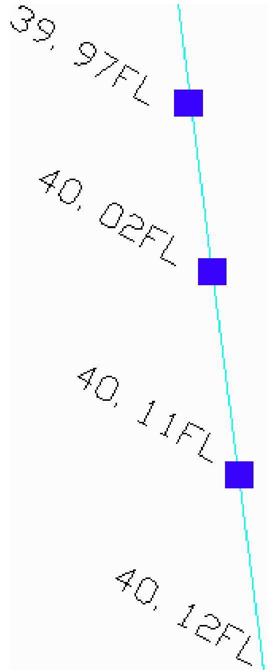


Keyboard Command: 2dto3dpt
Prerequisite: A polyline and points

2D to 3D Polyline-By Text

This command allows you to change 2D polylines to 3D polylines by elevation labels. This command will prompt you for samples of the elevation labels and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the polylines with their labels you want to convert.

You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 539.97, 540.02, 540.11 sometimes, like in the example on the side, they are listed as 39.97, 40.02, 40.11. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. This command will assign elevations from the labels to nearby vertices. If vertices do not have a close elevation label than they will be interpolated from vertices that are nearby elevation labels.



Prompts

Select sample of elevation text: Pick a text label
Select sample of a polyline to convert: Pick a polyline
Select polylines to convert and elevation labels.

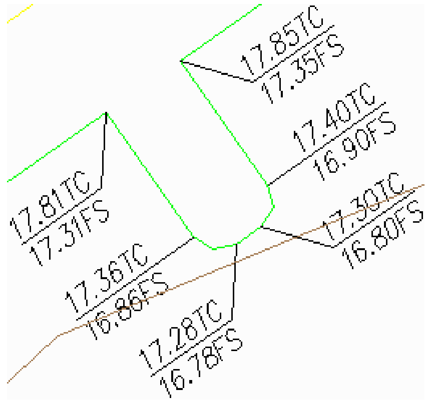
Select objects: Select all the entities to process
19 found, 19 total

Enter elevation to add to label values <0.00>: 500
Pre-processing entity #19 of 19
Filtering text entities
Processing elevation text #18
Remaking polyline #1

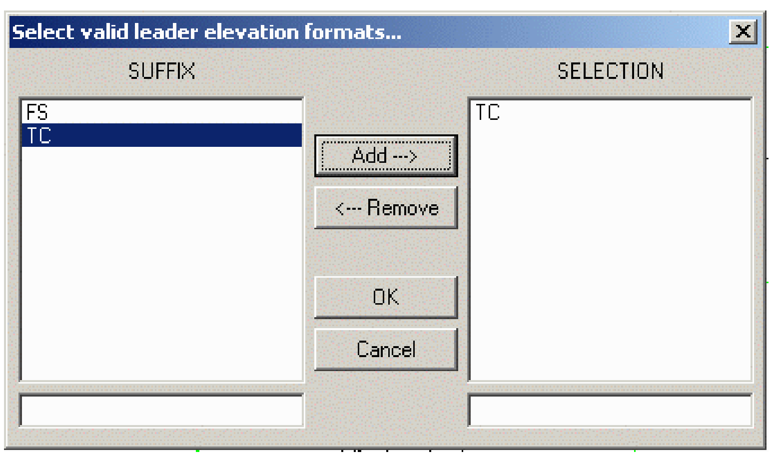
Keyboard Command: elevfb
Prerequisite: 2D polyline and elevation labels

2D to 3D By Text With Leader

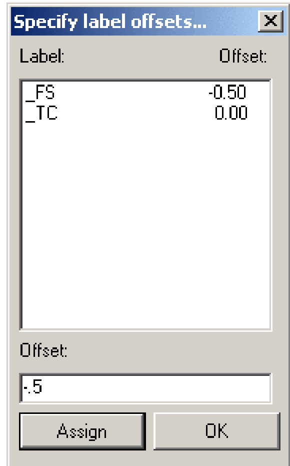
This command will assign elevations from the labels to the polylines by following the label leaders to their corresponding vertices on the polyline.

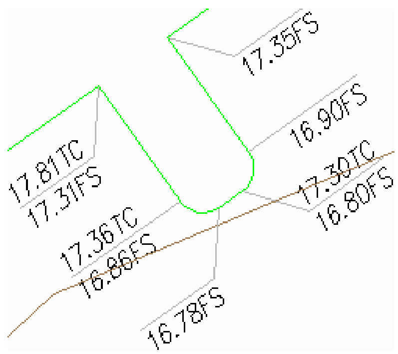


This command will prompt you for samples of the elevation labels, the leaders, and the polylines to convert. The program uses these samples to know the layer names for the labels and linework to process. Then select all the labels and leaders for the polylines you want to convert. You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 800s instead of labeling every elevation 817.85, 817.40, 817.30 sometimes, like in the above example, they are listed 17.85, 17.40, 17.30. This command allows you to add a given amount, such as 800, to every label elevation to produce the correct elevation in the drawing.



Carlson TakeOff searches for all leaders and gathers their associated text. If the program finds different labels in the elevation text, then this dialog box allows you to select the text you want to create 3D polylines. In this example you might want to use elevations followed by TC. This dialog box allows you to select that text and exclude the other text which is not to be used in the elevations of the polyline, such as FS.





If you are creating 3D polylines from multiple elevation labels than this dialog box will allow to offset certain labels by a given amount. In the above example you can offset an elevation labeled FS by .50 so that it matches vertices set by TC labeled elevations.

Prompts

Select sample of elevation text: Pick a text label
Select sample of an annotation leader: Pick an annotation leader
Select sample of a polyline to convert: Pick a polyline
Select polylines to convert, leaders and elevation labels to process.
Select objects: Select the desired entities
22 found
3 were filtered out.

Select objects:

Enter elevation to add to label values <0.00>: 800
Pre-processing entity #19 of 19
Filtering text entities
Processing leader #6
Remaking polyline #1

Keyboard Command: elevfl
Prerequisite: 2D polyline, elevation labels, and leaders

2D to 3D Polyline by Start-End Elevations

Function

This command allows you to convert a 2D polyline to a 3D polyline by specifying the starting and ending elevations of the polyline. All intermediate polyline vertice elevations are linearly interpolated from these end point elevations.

Prompts

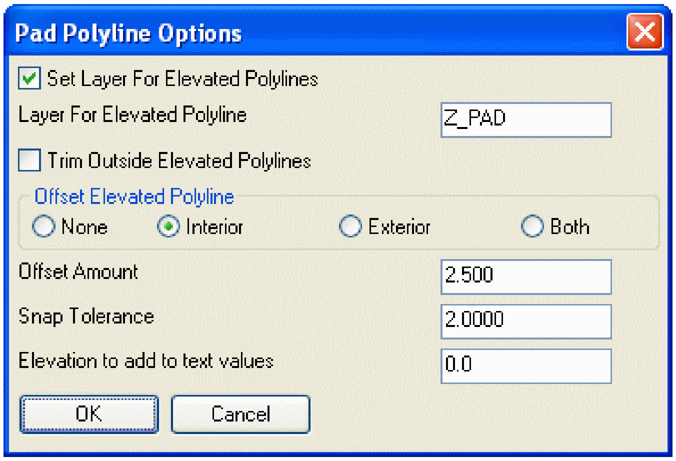
Select polyline to assign elevations:
Enter starting elevation: *109.85*
Percent/Ratio/<Enter ending elevation>: *112.16*
Select polyline to assign elevations (Enter to End): *press enter to end*

Keyboard Command: 2dto3dpl
Prerequisite: A polyline

Pad Polyline By Interior Text

This command allows you to set one or more pad elevations using interior text labels.

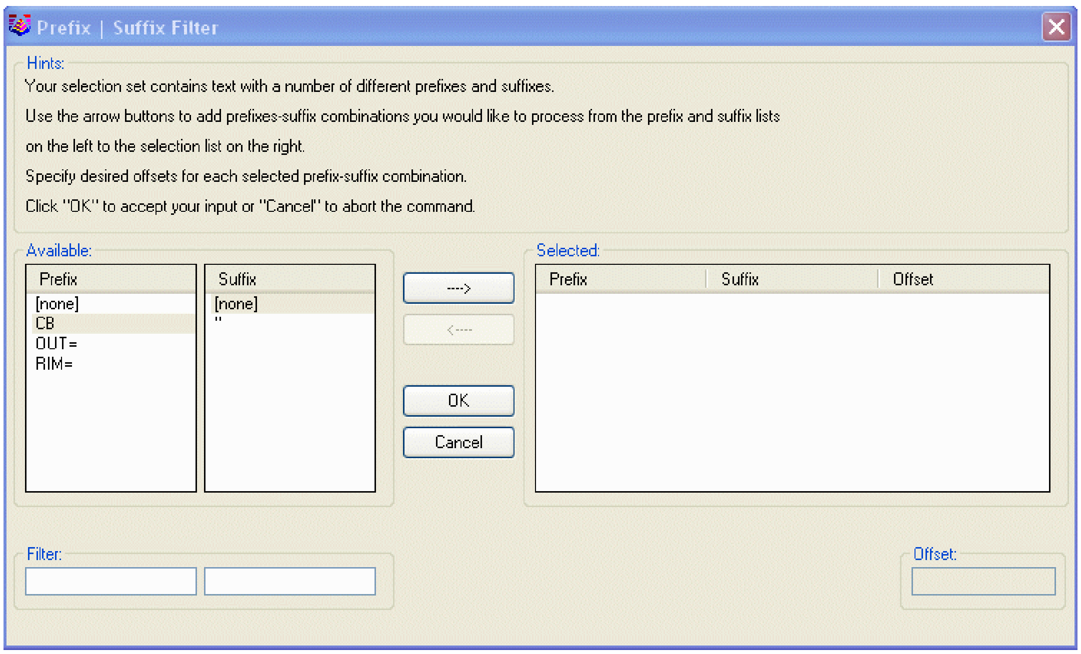
After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundary of the pad. Sometimes pads are drawn with linework from two different layers and Carlson TakeOff allows you to pick all the correct linework.



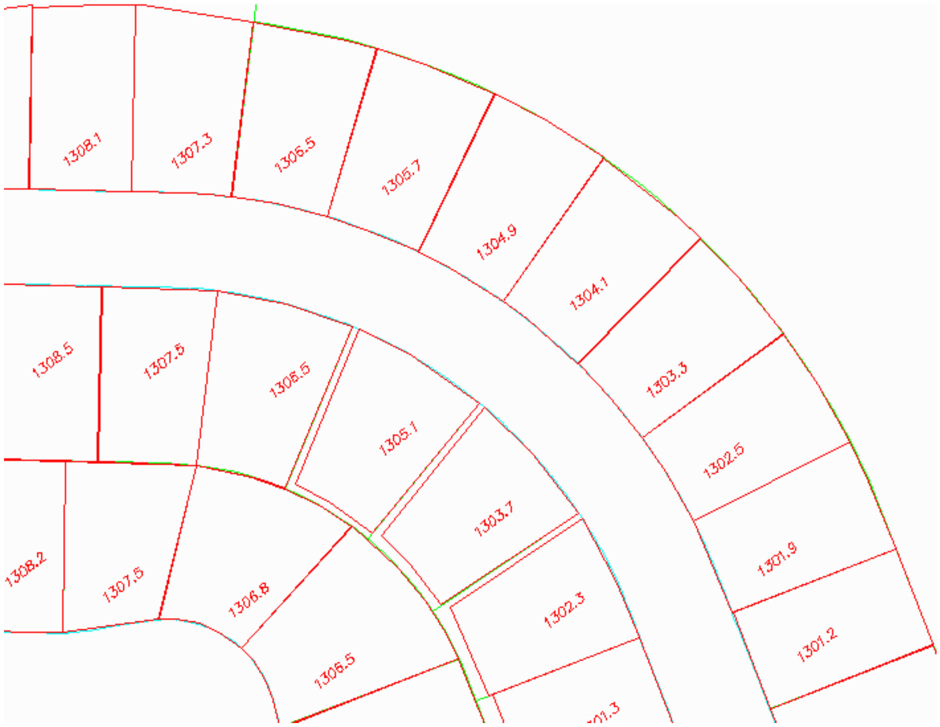
This dialog box allows you to create a new layer with the correct x,y coordinates and elevations. If the pad shares the same coordinates with other linework with different elevations than this dialog box allows you to offset the new polyline to avoid the problem of shared occupied points with different elevations. You can choose to have an interior offset or an exterior offset and also decide how much to offset the new polyline. Selecting Both will give

both the interior pad elevation and the exterior contour elevations. This helps the transition from you pad elevation to the design contouring. The Snap Tolerance field joins linework which falls within the range you set to create a pad. Trim Outside Elevated Polylines will trim out contour elevations that go through your pad that you are not using elevations from within the pad.
Elevation to add to text values adds to the values from the elevation labels. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing.

After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundary of the pad. Sometimes pads are drawn with linework from two different layers and Carlson TakeOff allows you to pick all the correct linework. In addition, if your text has multiple Prefixes and Suffixes you will be prompted to select the ones you want to use the elevation from.



After clicking <OK> select all the pads and their elevation labels that you wish to change, press <Enter>, and the new layer with elevations will be created and placed in the Design target.



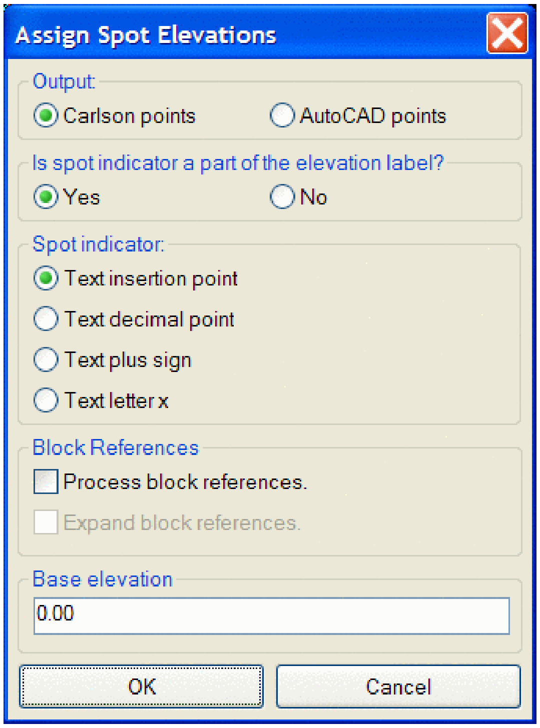
Prompts

Select layer sample of elevation text: Pick a label text
Selected text layer —-TX07
Select layer sample of boundary linework:
Selected linework layer PAD
Select another layer sample of boundary linework (Enter to continue):
Select text and linework to process.
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects:
Analyzing entire selection...
Set elevation for 1 polylines.

Keyboard Command: pad_by_text
Prerequisite: Pad polylines and elevations

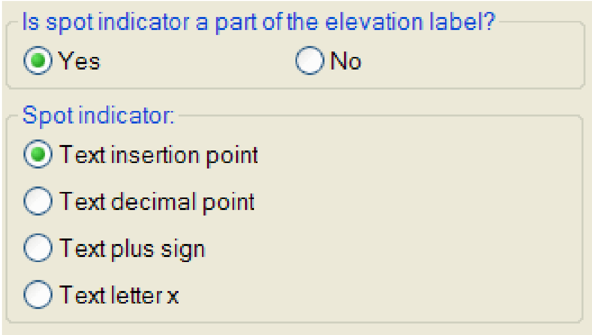
Convert Spot Elev To Points

This command takes spot elevation entities with zero elevations and assigns them elevations according to corresponding elevation labels. This dialog box allows you to choose the format of the spot elevations entities that you want to convert.



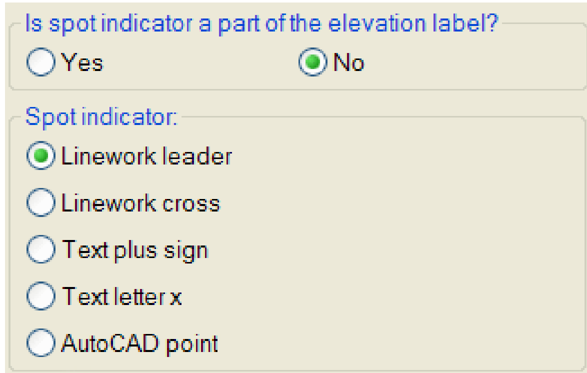
Output:
Carlson points: creates Carlson points at elevation of spot and stores them in coordinate file
AutoCAD points: creates AutoCAD point objects at elevation of spot

Is spot indicator a part of the elevation label?
If set to "Yes", four choices for Spot indicator are available to select from:

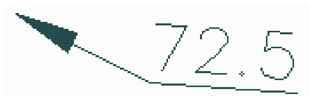


Text insertion point: uses the insertion point of the text for the location of the new point
Text decimal point: uses the decimal point in the text for the location of the new point
Text plus sign: uses the plus sign in the text for the location of the new point
Text letter x: uses the letter x in the text for the location of the new point

If set to "No", five choices for Spot indicator are available to select from:



Linework leader: creates a data point at the end of a leader

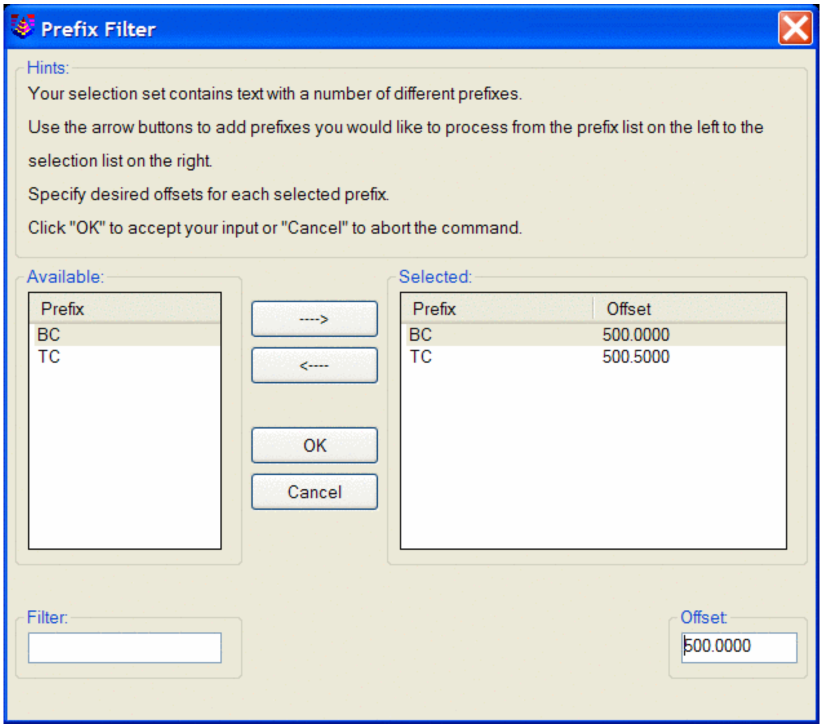


Linework cross: creates a data point at the intersection of a linework cross
Text plus sign: creates a data point at the insertion point of a text plus sign
Text letter x: creates a data point at the middle of a text letter x
AutoCAD point: creates a data point at the node of an AutoCAD point

Block References:
Process Block References: If check box is cleared, Carlson Civil searches only text entities for elevations, but if checked, Carlson Civil will search block references for elevations that are stored as attributes of a block. Use this option if the elevation is an attribute and the symbol designating the location of the spot elevation are both part of the block definition.
Expand Block References: Use this option to search block references when the elevation is stored as an attribute of a block, but the symbol designating the location of the spot elevation is a different block or even other geometry that is not defined within a block.

Base elevation: The value entered here is added to the existing spot elevations for all newly created points. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. Note: The base elevation will not be added to any elevations that are closer to the base elevation value than they are to 0; e.g. if a base elevation of 500 is specified, 500 will be added to elevations like 23.4, 45.5, etc, but will not be added to elevations like 456.4 or 468.9.

Prefix Filter: Carlson Civil examines all selected spot elevations for prefixes or suffixes. If they are all the same, the command proceeds, but if there are different prefixes and/or suffixes found, the Prefix Filter dialog box is invoked. This dialog box allows you to select which prefixes and/or suffixes to use to create spot elevations, and also allows you to use different offset values for each. .



Prompts

Starting point number <1>: *press Enter*
Select TEXT, MTEXT spot elevations to process and any associated leader lines:
Select objects: *pick entities to process*
Pre-processing entity #40 of 40...
Filtering text entities
Processing elevation #40...
Converted 40 spot elevations.

Keyboard Command: spotelv2
Prerequisite: Spot elevations

Set Point Elevations by Surface Model

This command assign elevations to points by surface model.

Prompts

Choose Grid or Tmesh file to process dialog *choose existing GRD, TIN or FLT file*
Select points from screen, group or by point number [<Screen>/Group/Number]? *press Enter*
Select points to elevate.
Select objects: *select points*

Keyboard Command: 3dpts_tin
Prerequisite: A surface model

File Name: \lsp\

Set Point Elevations by 3D Polylines

This command assign elevations to points by referencing 3D polylines.

Prompts

Maximum Offset Tolerance <1.0>: *press Enter*
Percent slope from reference polyline <0.0>: *.2*
Vertical Offset <0.0>: *5*
Select points from screen, group or by point number [<Screen>/Group/Number]? *press Enter*
Select points to elevate. *select point*

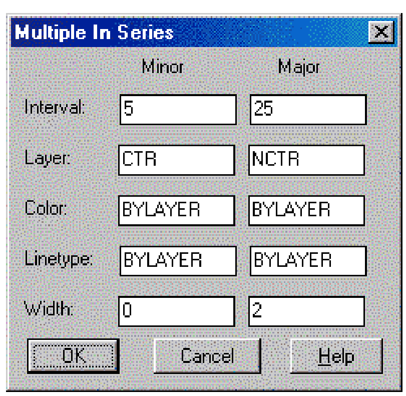
Keyboard Command: 3dpts_3dp
Prerequisite: 3D polylines
File Name: \lsp\

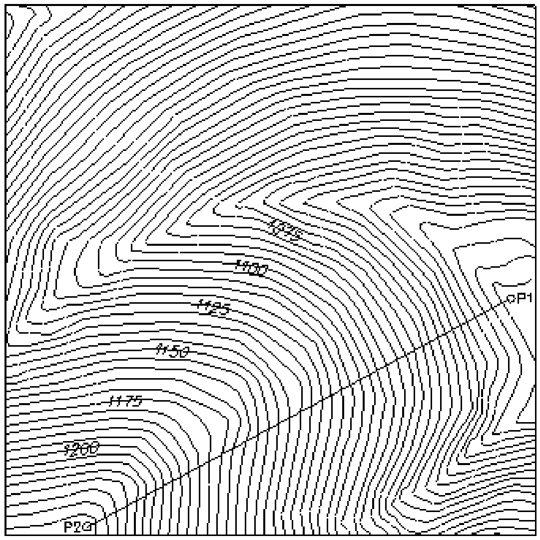
Assign Contour Elevation - Multiple in Series

This command can be used to quickly and accurately assign the elevation of series ofAutoCAD polylines that have been converted from raster or digitized without correct elevations. Theroutine will automatically assign elevations to the polylines crossing the fence line selected by two points.At the same time the elevations are changed, the program can assign it a new layer, color,linetype, and polyline width. This process usually works best if contours are in a temporary (white)layer to start. When they are processed, they will take on the color of the new layers making it easyto distinguish which polylines have been processed.

Prompts

Settings/First Point: *(Press S to change settings or pick first point.)*
Second Point: *(Pick second point)*
Beginning Elevation <0.00>: *1020*
Increment Direction U/D <U>: *(enter)*

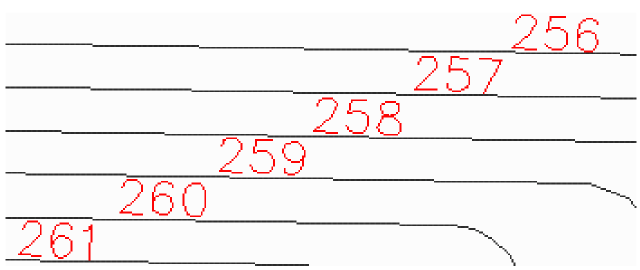




Keyboard Command: grpcelev
Prerequisite: digitized polylines

Assign Contour Elevation - From Contour Labels

This command allows you to set elevations to contours from elevation labels.



Select a sample of the elevation text to be used on the contouring. Next, select a sample of the contouring that you want to add the elevations to. Now select all the contours and their corresponding elevation labels and press <Enter>. Carlson TakeOff will then add elevations to all the contours. You may be prompted to distinguish what contour goes with what elevation label. You can either press <Enter> to accept the contour that Carlson TakeOff has selected or you can Press <N> to choose another contour.

Prompts

Select sample of elevation text:
Select sample of a contour line:
Select contour lines and elevation text to process.
Select objects: all

5049 found
4041 were filtered out.

Select objects:

Joining adjacent polylines...
Reading the selection set ...
Joining ...
Pre-processing entity #1008 of 1008
Filtering text entities
Processing elevation text #518

Conflict detected: pick contour corresponding to current elevation text
Press N for next selection or Enter to accept current:
Remaking polyline #311

Keyboard Command: TXTCELEV
Prerequisite: contours and contours labels

Assign Contour Elevation - Single Elevation Group

This command changes the elevations of polylines and can be used to set the elevations of contour polylines. The routine begins at a specified elevation and prompts for a selection set of polylines to set to the elevation. Then the routine repeats using the last elevation plus the elevation increment. Enter an empty selection set to exit the routine.

Prompts

Starting elevation <0.0>: *500.0*
Contour interval (negative for down) <1.0>: *5.0*
Select polylines to set to elevation 500.0.
Select objects: *pick the polylines*
Select polylines to set to elevation 505.0.
Select objects: *pick the polylines*
Select polylines to set to elevation 510.0.
Select objects: *Press Enter*

Keyboard Command: setcelev
Prerequisite: polylines

Drape 3D Polyline On Surface

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge breakline could be generated from contour lines by drawing a 2D polyline along the ridge and across the contours. Then this command could grab the contour line elevations along the polyline to make a ridge breakline.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in triangulation (.flt or .tin) file. If you use a file, then you can also use the polyline's current elevation as a vertical offset from surface.

Prompts

By Screen Entities:
Source of surface model [File/<Screen>]? *Type S for Screen*
Select polylines to convert.
Select objects: *select the polyline(s) to convert*
Select surface 3DFaces, lines and polylines.
Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Reading points ... 692

Keep existing polylines [Yes/<No>]? *Press Enter*
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.
Set layer name for converted polylines [Yes/<No>]? *Press Enter*
This allows you to assign the new polyline to a layer.
Converting polylines ...
Converted 1 polylines.

By a .flt or .tin File:
Source of surface model [<File>/Screen]? *Type F for File*
Select polylines to convert.
Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Use current polyline elevations as vertical offset from surface [Yes/<No>]? *Press Enter*
This will offset the new polyline by its current elevation. That is, if a polyline has an elevation of -4 and the surface you are converting it to has an elevation of 800, then saying Yes will drape the polyline at an elevation of 796.
Keep existing polylines [Yes/<No>]? *Press Enter*
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.
Set layer name for converted polylines [Yes/<No>]? *Press Enter*
This allows you to assign the new polyline to a layer.

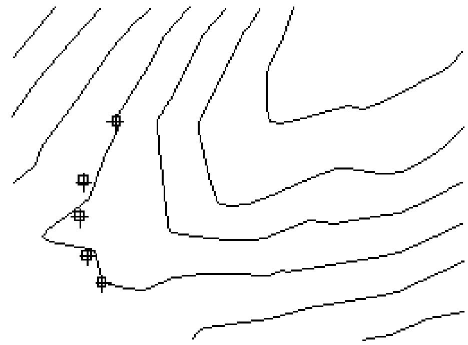
Keyboard Command: 2dto3dp
Prerequisite: A polyline and surface lines or grid file or triangulation file.

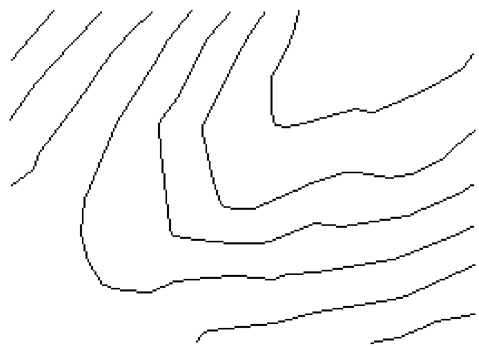
Edit Contours

This command revises a segment of a contour polyline. Begin by picking a point on the contour where you want to start editing. Then pick new points for the polyline. When finished picking new points, press Enter and then pick a point on the contour to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

Prompts

Select contour to edit: *pick the contour polyline at the place to start editing*
Pick intermediate point (Enter to End): *pick a point*
Pick intermediate point ('U' to Undo, Enter to End): *pick a point*
Pick intermediate point ('U' to Undo, Enter to End): *Press Enter*
Pick reconnection point on contour: *pick the contour polyline at the place to join*





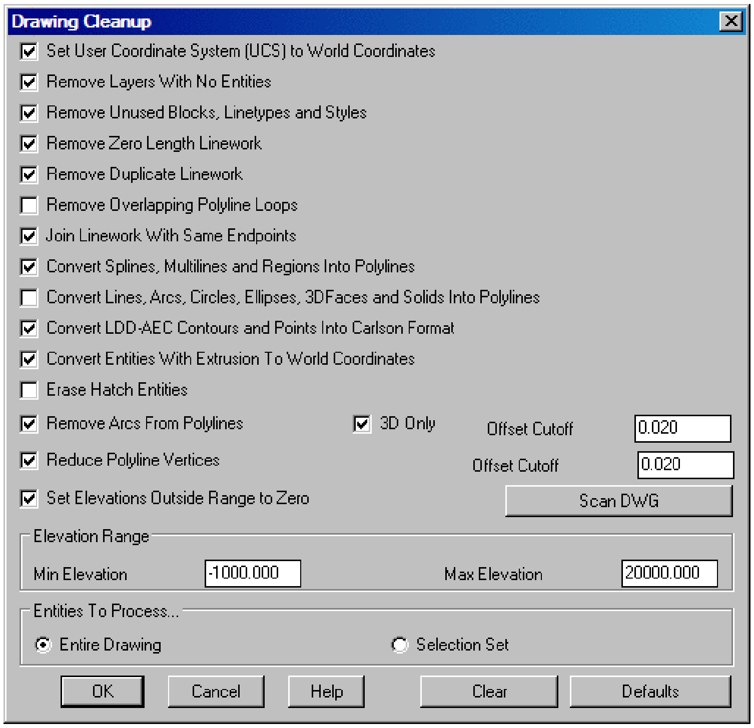
Edit this contour by picking new points
Contour with segment replaced with new points

Keyboard Command: editctr
Prerequisite: polylines with elevation (contour polylines)

Takeoff Menu 8

Drawing Cleanup

The Drawing Cleanup dialog box allows you to perform many functions that fix common errors, and it removes unnecessary data found in many drawing files. It also converts incompatible data into useful entities. This command offers many filters that audit the drawing file and allows you to select which options and settings you want to use. A report of the cleanup results will be displayed upon completion. Always save your file when the drawing cleanup routine is complete.



Set UCS to World Coordinates

This sets the UCS (user coordinate system) to the world coordinate system (WCS). Carlson works exclusively in the world coordinate system and there is no way to change this setting. In AutoCAD, it is possible to change the coordinate system from WCS. If you receive a drawing in which the coordinate system is not set to world, click this on to restore the UCS.

Remove Layers With No Entities

AutoCAD drawings work with a "BYLAYER" concept meaning that layer definitions define the drawing. For example, the layer named EOP might be used to display polylines at the Edge Of Pavement in the drawing. Many times extra layers get defined by a user but not used to display any objects. This function removes any layers defined in the drawing that are not being used.

Remove Unused Blocks, Linetypes and Styles

This functions removes this unused information from the drawing.

Remove Zero Length Linework

This function seeks out and removes any linework definition that have zero length. Point nodes are not removed.

Remove Duplicate Linework
This function finds any duplicate linework in the drawing and removes all but one set.

Remove Overlapping Polyline Loops
Polylines that completely overlap themselves are broken into two different polylines.

Join Linework With Same Endpoint
This function finds common endpoints on linework on common layers with common elevations and joins the linework into a continuous polylines. This is very helpful for future selection sets.

Convert Splines, Multilines and Regions Into Polylines
Some CAD applications utilize Spline Object Definitions and Regions, Carlson utilizes basic polyline/polygon definitions. This function finds any Splines and/or Regions defined in the drawing and re-defines them as simple polylines or polygons.

Convert Lines, Arcs, Circles, Ellipses, 3DFaces and Solids Into Polylines
By converting Lines, Arcs, Circles, Ellipses, 3D Faces, and Solids into Polylines, you can use the variety of Polyline commands available in Carlson.

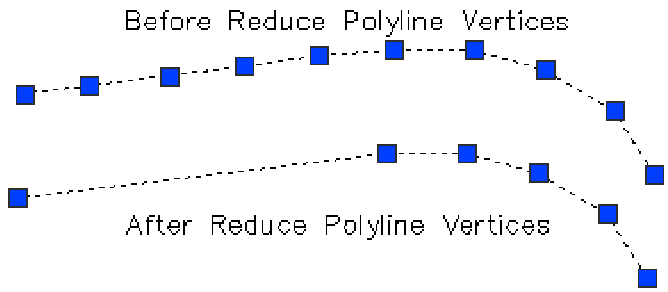
Convert LDD-AEC Contours and Points Into Carlson Format
Drawings created in the Land Development Desktop CAD program can contain special objects known as LDD-AEC contours that define their topographic contour display. This function locates those special objects and re-defines them as simple 2D polylines retaining their elevation values.

Convert Entities With Extrusion To World Coordinates
Drawings created in the Land Development Desktop CAD program can contain special objects known as LDD-AEC contours that define their topographic contour display. This function locates those special objects and re-defines them as simple 2D polylines retaining their elevation values.

Erase Hatch Entities
Carlson offers many hatch display options, however hatch entities have no 3D value. This function removes all hatch entities in the original drawing to help reduce the size and clutter of the drawing file.

Remove Arcs From Polylines - Offset Cutoff
This function replaces arcs in polylines with a series of short chord segments. The purpose is to prepare the polylines for modeling since arcs need to be converted into segments to be part of the triangulation model. The density of chord segments is controlled by the offset cutoff. This cutoff represents how much the polyline can move horizontally. A smaller cutoff will result in more chord segments. The option for 3D Only controls whether only polylines at zero elevation or both zero and elevated polylines get processed. Sometimes you may want to leave the arcs in zero elevation polylines when these polylines represent road alignments and are not part of the surface model.

Reduce Polyline Vertices - Offset Cutoff
This function utilizes a pre determined offset amount and removes unnecessary polyline vertices that fall within the offset amount.

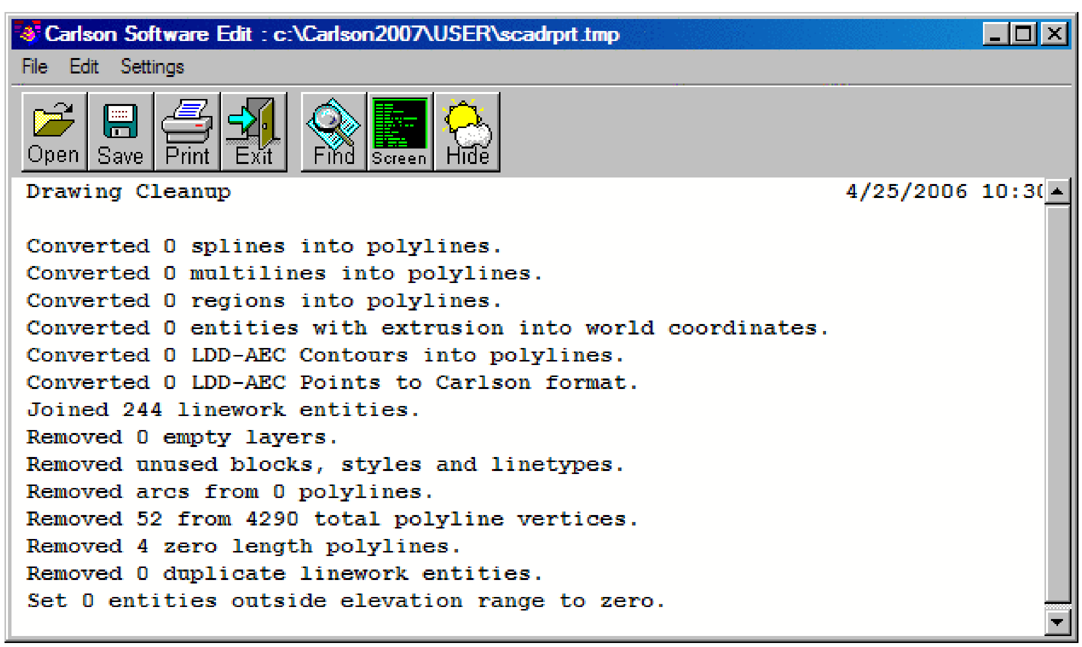


Set Elevations Outside Range to Zero and Elevation Range
This function comes with a "Scan DWG" option that audits the elevation range in the drawing file. Once the minimum and maximum elevation range has been set, manually or by a scan, all objects that fall outside the set range are moved to elevation zero. All objects at zero elevation do not contribute to the 3D model.

Entities To Process...
This allows you to run the command for the entire drawing or for a selected set.

Default
This allows you to return to the Carlson Drawing Cleanup default settings.

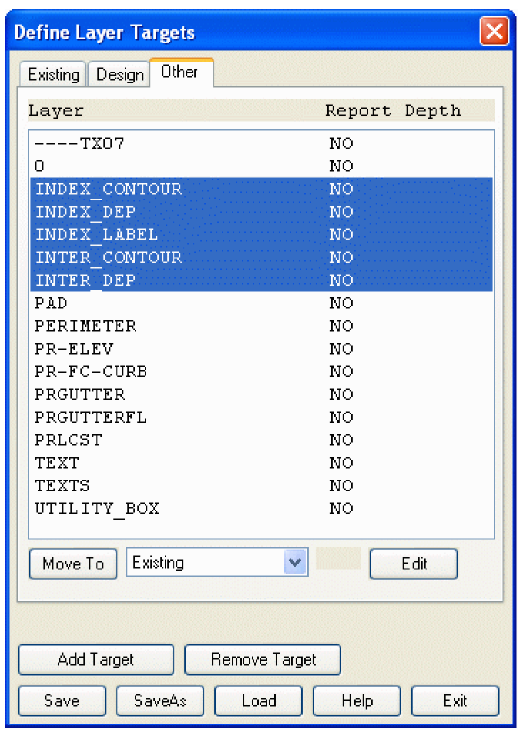
Final Report
This example report displays the results of drawing cleanup. Like all reports in Carlson, this report can be saved to a text file, sent directly to your printer, or pasted onto the screen ad text entities.



Pulldown Menu Location: File
Keyboard Command: dwg_cleanup
Prerequisite: None
File Name: \lsp\poly3d.arx

Define Layer Target/Material/Subgrade

The Define Layer Targets dialog box (shown here) offers many functions that will ultimately make up the surface models used in volume and material calculations. Every entity (line, polyline, point, etc) in a drawing is assigned a layer name. Carlson Takeoff uses the entity layer names to define which entities represent the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces.The drawing entities are assigned their target surface by their layer name. For example, if polylines representing design contours are on the layer "Final", then "Final" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other. The Define Layer Targets dialog has three lists for layer targets: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog. To move a layer to a target destination, highlight the desired layer, choose the target from the Move To list and pick the "Move To" button. All layers populate the "Other" target before being assigned to "Existing" or "Design".

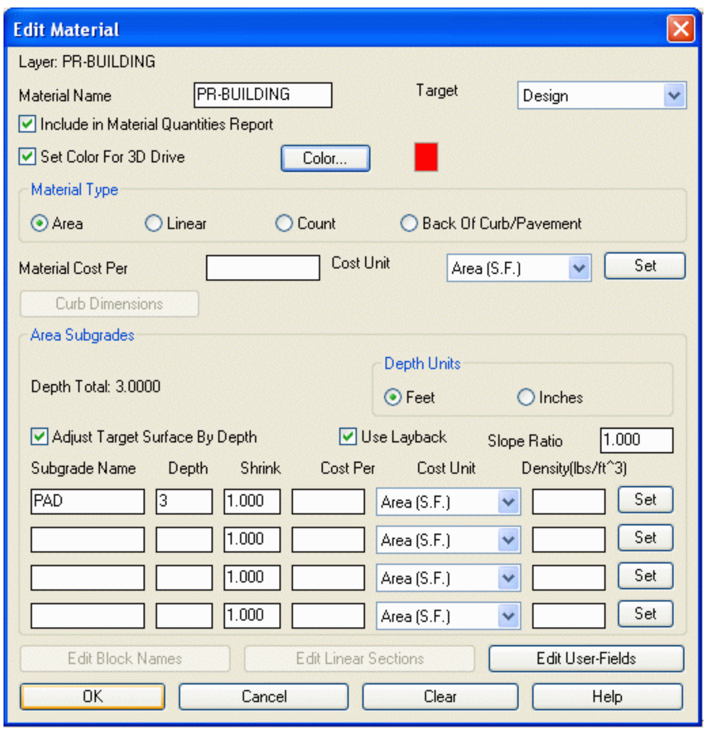


Besides the basic three layer targets (Existing, Design and Other), you can add more target surfaces with the Add Target button. When another target is defined, there will be another tab along the top of the Define Layer Targets dialog. Then layers can be assigned to this additional target surface. The only pre-defined additional surface is Overexcavate. The layers that are assigned to the Overexcavate target can be modeled into the Overexcavate

surface using the Make Overexcavate Surface command. Besides Overexcavate, the other additional targets are user-defined. The layer targets can be modeled using the Make User-Defined Surface command. Then these surfaces can be used in Takeoff commands by assigning them to a Takeoff existing or design surface using the Set Active Surfaces command.

Edit Materials

The "Edit" button activates the Edit Material dialog box (shown here) and allows you to define the Material name and Subgrade depths and names. Besides assigning target surfaces by layer, layers are also used to define material names and subgrades depths. By assigning a material name, Subgrade names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface, or tie into the design surface by a Slope Ratio if "Use Layback" is checked on. **The polylines on the layer used for a Materialmust be closed polylines.** Carlson Takeoff supports nested Subgrade polylines for exclusion areas such as islands by counting how many Subgrade polylines surround an area. If the number is odd, then the area is included in the Subgrade. The even count regions in the area are not part of the Subgrade. To activate the Edit Material, select a layer from the list and then choose "Edit".



Include in Material Quantities Report

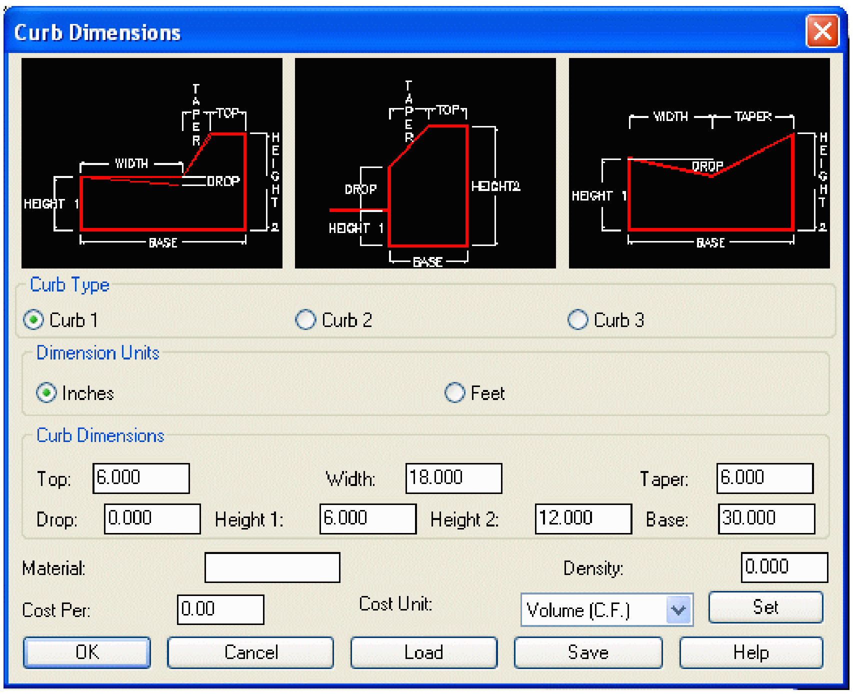
With this option checked on, the material that is named will appear in the Material Quantities Report. The report will include either the area of the material, the linear length of the material, or the number of items counted on the layer defining the material. This is accomplished by choosing "Area", "Linear", or "Count" for the Material Type.

Set Color For 3D Drive

This options checked allows you to assign a color for this particular material for display purposes during the 3D view/drive simulator.

Material Type

This will report the subgrade by area, linear length, count, or as curb area. If you choose Back of Curb/Pavement then you can pick on the Curb Dimensions button and bring you to the below dialog:



This option will calculate your curb volume as well as act as the limit of the pavement. The pavement limit will be from the Back of Curb polyline offset by the length of the Curb base. In the above case the base is 30 inches wide. Therefore, the pavement area will stop 30 inches before the Back of Curb polyline.

Material Cost Per Cost Unit

Use this field to add the value of the multiplier for the unit cost of your material. If the material type is an area that has multiple subgrades, use the available fields below to add each individual subgrade name, depth and cost value per unit type. If a linear or count type material type option is selected, use the "length in feet", or the "count" unit options.

Adjust Design Surface by Depth

This determines whether the subgrade depths are incorporated in the design surface or not.

Area Subgrades

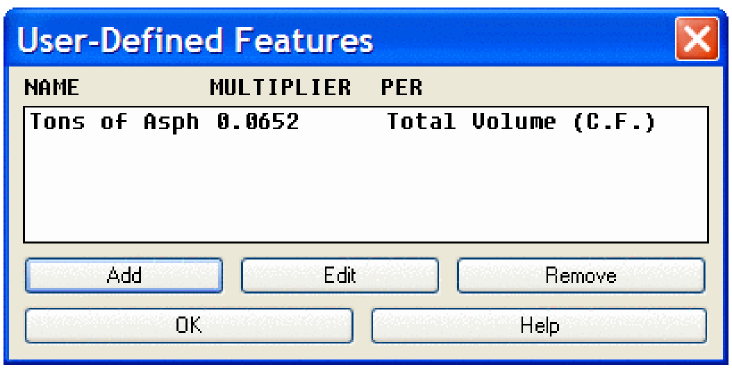
Depth Units

Select the "feet" or "inches" as the unit value desired for depth of subgrades.

Subgrade Name Depth Shrink Cost Per Cost Unit Density

Use these options for areas that are represented with a single/multiple closed polygon/polygons in the drawing, but have multiple material types defining the surface. Simply name each "lift" in the area, issue a depth value and add a cost unit if desired, or click on select and choose a material from the Materials Library (see Define Materials Library for more). Carlson Takeoff will report each subgrade material value in the material quantities report. The Shrink factor is multiplied by the subgrade volume in the material quantities report and represents the fill shrinkage. A Density factor can be entered in when using Cost Per Tons.

If user entered values are needed in the report use the "Edit User-Fields" button to activate the "User Defined Features" dialog box shown here. Choose the "Add" button to define needed fields such as TONS of material or BAGS OF GRASS SEED for reporting options.



Once all of the material subgrades, depths and cost units or user defined units have been defined, select save to preserve the settings in a .trg file, the "save as" function allows the user to name the file to load later.

Prerequisite: none
Keyboard Command: define_tk_layers

Edit Selected Layer

Use this command to click on any layer and advance to the Edit Materials dialog from the Define Layer Target/Material/Subgrade command.

Prerequisite: none
Keyboard Command: edit_tk_layer

Set Layer For Existing

Set Layer For Existing allows the user to pick the layers from objects on the screen and assign them to the Existing Layer.

Prerequisite: none
Keyboard Command: set_existing_layer

Set Layer For Design

Set Layer For Design allows the user to pick the layers from objects on the screen and assign them to the Design Layer.

Prerequisite: none
Keyboard Command: set_design_layer

Set Layer For Other

Set Layer For Other allows the user to pick the layers from objects on the screen and assign them to the Other Layer.

Prerequisite: none
Keyboard Command: set_other_layer

Boundary Polyline

The Boundary Polyline options allow the user to Set the Boundary Polyline, Set the Exclusion Polylines, Clear Exclusion Polylines, Hatch the Boundary Area, Erase the Boundary Hatched area.

Set Boundary Polyline

Use this command to select the "CLOSED" polyline that defines the outer most limit of the disturbed area. This boundary should dissect the site at the point where the design contours meet the existing contours, or where the limit of work will occur. If your site contains separated areas (such as different phases or isolated sections of work), then multiple Boundary Polylines can be used. Volume calculation will take place inside this boundary.

Prerequisite: a closed polyline
Keyboard Command: tag_inclu

Set Exclusion Polylines

Use this command to select the "CLOSED" polylines the define the areas inside the Boundary Polyline that will not be disturbed. These boundaries should also be at the intersection of the proposed and existing surface. A pond or wetland that will not be removed during construction is a good example of an Exclusion Area.

Prerequisite: a closed polyline
Keyboard Command: tag_exclu

Clear Exclusion Polylines

Use this command to select polylines that were previously defined as exclusion polylines but are no longer needed as exclusion areas.

Prerequisite:
Keyboard Command: untag_exclu

Highlight Boundary Polylines

This command highlights the polyline you set as the Boundary Polyline.

Prerequisite: a boundary polyline
Keyboard Command: highlight_boundary

Hatch Boundary Area

Use this command to confirm the boundary polylines that have been selected are correct. This hatched area can also be utilized in exhibits of the drawing.

Prerequisite: a boundary polyline
Keyboard Command: hatch_boundary

Erase Boundary Hatch

This command erases the hatch drawn in the plan view.

Prerequisite: a boundary hatch
Keyboard Command: erase_boundary

Areas Of Interest

Areas of Interest can be used to calculate volumes and material quantities within a specified area. The Area Of Interest perimeters are defined by user-selected closed polylines and each area is assigned a name. The Area Of Interest polylines can be assigned either as inclusion or exclusion perimeters for the area. You can have any number of exclusion perimeters within an inclusion but inclusion perimeters cannot be inside exclusions.

The Areas Of Interest (AOI) commands allow you to Tag/Untag Areas of Interest, Identify/Report Areas of Interest and Hatch/Label Areas of Interest.

Tag Area Of Interest

This command allows the user to select polylines and exclusion perimeters that define phases of a project. Carlson Takeoff will separate each area of interest in the volume and material reports.

Prerequisite: a desired polyline
Keyboard Command: tag_aoi

Area Of Interest by Interior Text

This command allows the user to select text from the screen to name AOIs and linework to determine the area.

Prerequisite: area linework and text
Keyboard Command: txt2aoi

Untag Area Of Interest

This command allows the user to remove previously tagged areas.

Prerequisite: an area of interest
Keyboard Command: untag_aoi

Identify Area Of Interest

This command allows users to identify AOI by either picking on a polyline(s) or by searching the entire drawing. The command will then report the AOI name, layer, type, starting point, and highlight the polyline in the plan view.

Prerequisite: an area of interest
Keyboard Command: id_aoi

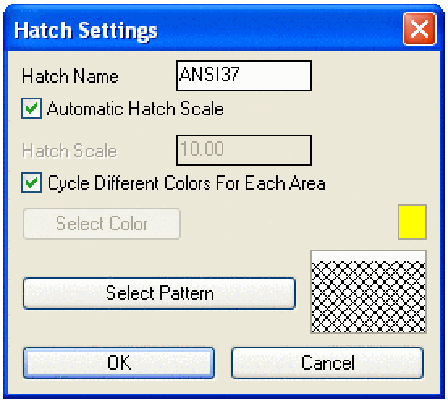
Report Area Of Interest Areas

Use this command to report the Inclusion or Exclusion area (SF), the name, the layer, and the starting point.

Prerequisite: an area of interest
Keyboard Command: report_aoi

Hatch Area Of Interest Areas

This command allows the user to visually see AOIs in the plain view.



This command draws a hatch with a specified color and pattern for the Areas of Interest. The purpose is to allow you to visually review AOIs to make sure that the area coverage is correct.

The command displays a dialog for the hatch pattern, color and scale. The scale determines how spread out the pattern is within the hatch. The Automatic Hatch Scale option checks the size of the subgrade areas and sets the scale to make the pattern fit. Cycle Different Colors For Each Area will give each AOI it's own color so that you can distinguish between different AOIs.

The resulting hatch areas show where the AOI is applied. Exclusion Areas of AOIs will not be hatched.

Prerequisite: an area of interest
Keyboard Command: hatch_aoi

Erase Area Of Interest Hatch

This command erases AOI hatching.

Prerequisite: hatched area of interest
Keyboard Command: erase_aoi_hatch

Label Area Of Interest Areas

This command labels the AOI name and area in the plain view.

Prerequisite: an area of interest
Keyboard Command: label_aoi

Erase Area Of Interest Labels

This command erases AOI labeling.

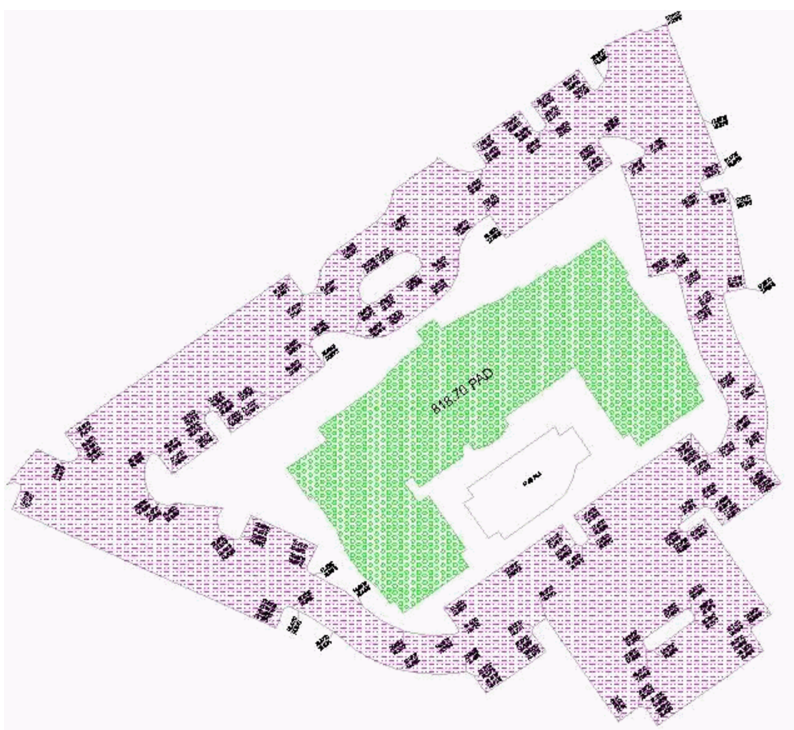
Keyboard Command: erase_aoi_labels
Prerequisite: hatched area of interest

Hatch Subgrade Areas

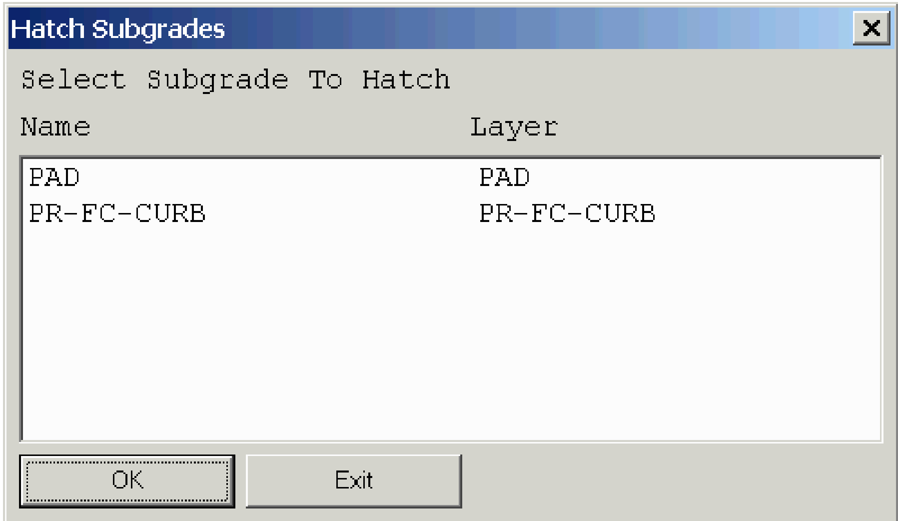
This command draws a hatch with a specified color and pattern for the area that the selected subgrade area applies to. The purpose is to allow you to visually review a subgrade area to make sure that the area coverage is correct.

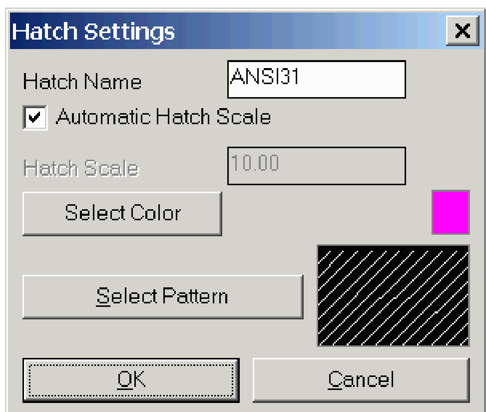
The command displays a dialog to select which subgrade to hatch. The list of available subgrades comes from the layers with subgrade depths as set in the Define Layer Target/Material/Subgrade command. Then there is a dialog for the hatch pattern, color and scale. The scale determines how spread out the pattern is within the hatch. The Automatic Hatch Scale option checks the size of the subgrade areas and sets the scale to make the pattern fit.

The resulting hatch areas show where the subgrade is applied. In the example below, notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Also note that the smaller pad area is not hatched because this polyline layer is different than the bigger pad polyline.



Prompts





Keyboard Command: hatch_subgrade
Prerequisite: subgrades

Erase Subgrade Hatches

This command removes from the screen the subgrade hatches created by the commandHatch Subgrade Area.

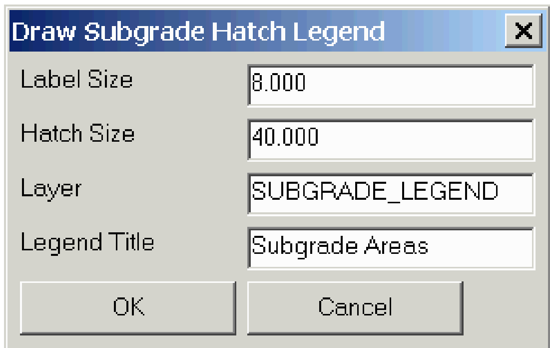
Keyboard Command: erase_subgrade
Prerequisite: hatch subgrade areas

Draw Subgrade Hatch Legend

This command draws a legend for the subgrade areas currently in the drawing. The legend includes the subgrade names and squares of the hatch patterns. The size of the labels, size of the hatch squares, layer for the legend

entities and the legend title are set in the dialog shown below. The subgrade hatches to include in the legend are automatically selected from all the subgrade hatches currently in the drawing that were created by the Hatch Subgrade Areas command.

Prompts



Keyboard Command: draw_subgrade_legend
Prerequisite: hatched subgrade areas

Label Subgrade Areas

Function

This command lets you label the subgrade depth and area (in sq. ft. or meters). The label is placed at the center of the subgrade area, but can be moved with the Move command under Edit.

Pull-Down Menu Location: Inquiry-> Subgrade Areas
Keyboard Command: label_subgrade
Prerequisite: Subgrade Areas

Erase Subgrade Labels

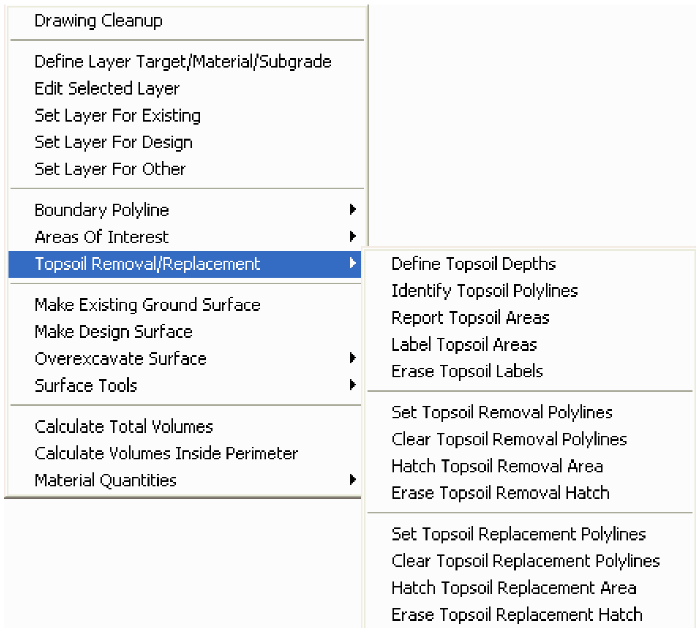
Function

This command erases subgrade labels.

Pull-Down Menu Location: Inquiry-> Subgrade Areas
Keyboard Command: erase_subgrade_labels
Prerequisite: subgrade labels

Topsoil Removal and Replacement

The Topsoil Removal and Replacement options (shown here) allow the user to Define Topsoil removal and replacement depths, Set topsoil removal and replacement areas by selecting closed polylines, Clear the selected boundary polylines if needed, Hatch the topsoil removal and replacement areas and Erase the hatched areas.



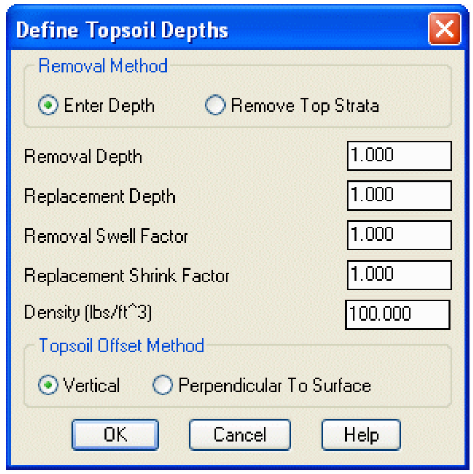
Define Topsoil Depths

This command requires user input to define the depth, or strata, of topsoil removal and replacement. Fill in the options available in the Define Topsoil Depths dialog (shown here). Carlson Takeoff will perform four functions with these values. First, the value set for the Removal Depth, or the Top Strata if selected, will be the "defined" removal amount from the Existing Ground Surface. Second, the calculated volume of topsoil removed will be included in the reporting options. Third, the value set for the Replacement Depth will be added "BELOW" the Finished Ground Surface model. Fourth, the amount of topsoil replaced will be included in the reporting options.

When topsoil depths are defined, the volume report routines will include the topsoil quantities. These topsoil quantities are in addition to the cut/fill for the existing to design surfaces for the site.

The Removal Swell Factor and Replacement Shrink Factor are multiplied by the topsoil removal and replacement quantities respectively in the volume report routines. The Density is used to report topsoil tons when the volume report option for tons is active.

The Topsoil Offset Method choose between offsetting the topsoil depth vertically or perpendicular to the surface. The perpendicular method will result in more topsoil quantities since it represents applying the topsoil depth to the slope area of the surface whereas the vertical method represents applying the topsoil depth to the horizontal area.



Prerequisite: topsoil depths
Keyboard Command: define_topsoil

Identify Topsoil Polylines

This command allows users to identify topsoil polylines by either picking on a polyline(s) or by searching the entire drawing. The command will then report the layer name and starting point for both removal and replacement polylines. These polylines are also highlighted in the plain view.

Prerequisite: topsoil polylines
Keyboard Command: id_topsoil

Report Topsoil Areas

Use this command to report the Inclusion or Exclusion area (SF), the type, the depth, the layer, and the starting point.

Prerequisite: topsoil areas
Keyboard Command: report_topsoil

Label Area Of Interest Areas

This command labels the topsoil type and area in the plain view.

Prerequisite: topsoil area
Keyboard Command: label_topsoil

Erase Area Of Interest Labels

This command erases topsoil labeling.

Prerequisite: hatched area of interest
Keyboard Command: erase_topsoil_labels

Set Topsoil Removal Polylines

Use this command to select the "CLOSED" polyline boundary defining the extents of topsoil removal and any "CLOSED" interior polylines that will not have topsoil removed. The outer boundary usually is the same polyline that defined the "Boundary Polyline" selected earlier. The internal polylines usually are those that have been selected as the "Exclusion Polylines". The layer names for these boundaries is irrelevant. You will be prompted to use the Removal Depth defined in the Define Topsoil Depths command or to customize your depth.

Prerequisite: polylines for removal
Keyboard Command: tag_topsoil_remove

Clear Topsoil Removal Polylines

This command allows the user to remove and previously selected Topsoil Removal Polyline boundaries.

Prerequisite: topsoil polylines
Keyboard Command: untag_topsoil_remove

Hatch Topsoil Removal Area

Use this command to display a hatch pattern over the entire area designated for topsoil removal.

Prerequisite: topsoil areas
Keyboard Command: hatch_topsoil_remove

Erase Topsoil Removal Hatch

Use this command to remove the hatch pattern that defined the topsoil removal area.

Prerequisite: hatched topsoil
Keyboard Command: erase_topsoil_remove

Set Topsoil Replacement Polylines

Use this command to select the "CLOSED" polyline boundary defining the extents of topsoil replacement, and any "CLOSED" interior polylines that will not have topsoil replaced. The layer names for these boundaries is irrelevant. You will be prompted to use the Topsoil Replacement amount defined in the Define Topsoil Depths command or to customize your amount.

Prerequisite: polylines for replacement

Keyboard Command: tag_topsoil_replace

Clear Topsoil Replacement Polylines

This command allows the user to remove and previously selected Topsoil Replacement Polyline boundaries.

Prerequisite: topsoil polylines
Keyboard Command: untag_topsoil_replace

Hatch Topsoil Replacement Area

Use this command to display a hatch pattern over the entire area designated for topsoil replacement.

Prerequisite: topsoil areas
Keyboard Command: hatch_topsoil_replace

Erase Topsoil Replacement Hatch

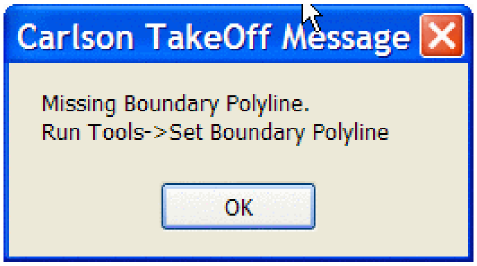
Use this command to remove the hatch pattern that defined the topsoil replacement area.

Prerequisite: hatched topsoil
Keyboard Command: erase_topsoil_replace

Make Existing Ground Surface

This command makes the triangulation models for the existing ground surface. There are three surfaces that are created: initial original ground, original ground after applying subgrade zones, and original ground after subgrade zones and topsoil removal. These surface files are automatically named as "filename-og.tin", "filename-ze.tin" and "filename-ex.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also, the file extension will be .tin for the binary format triangulation and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Existing surface, then the original ground after subgrades surface with be the same as the initial original ground surface. The topsoil removal depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil removal areas will lower the ground surface by the topsoil depth. If there aren't any topsoil removal areas, then the original ground after subgrade and topsoil surface will be the same as the original ground after subgrade surface.

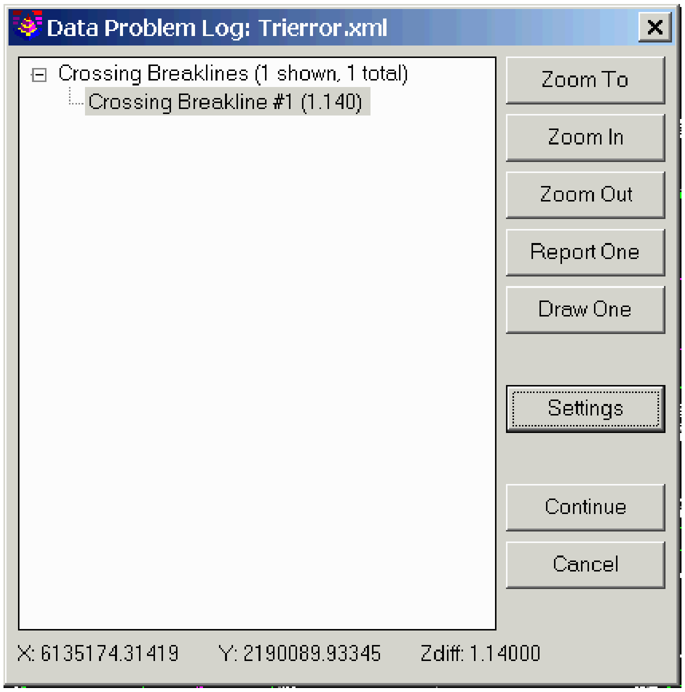
Before running this command, the layer names for the entities on the Existing layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site. If the boundary has not been set, the following error message will appear.



If this error message appears, run the "Set Boundary Polyline" command and pick the CLOSED polyline representing the boundary of the site.

When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Three types of conflicts are reported: Crossing Breaklines, Vertical Edges, and Breakline T-Intersections. Crossing Breaklines indicates that the intersection of two entities does not have a common elevation. Vertical Edges indicates that two entities or vertices of differing elevations have the same x-y location, thus forming a vertical plane. Breakline T-Intersections indicates that a 3d entity is abutting another entity, but the second entity doesn't have a vertex at the point of intersection. Each type of conflict is listed in its own category.

The Data Error Log shows the amount of elevation difference at each error. You can use the Data Error Log to review, report and draw markers at these error locations. Then you can exit the Data Error Log and fix the data errors with the commands in the Elevate menu or other drafting tools. After these errors are fixed, you can run Make Existing Ground Surface again.



Clicking to the "plus" sign beside a category will display the individual conflicts within that category. When a line item error is selected, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Zoom functionality allows the user to more closely inspect the specific problem area, and if needed a marker can be drawn or a report generated for an individual conflict or conflicts.

Zoom To pans the drawing to move the selected conflict to the center of the screen. The zoom functions are only active when a single line item is selected.

Zoom In zooms in on the highlighted area for closer inspection**.** Multiple picks on the zoom button will increase the magnification.

Zoom Out zooms out away from the highlighted area.

Report All/One toggles between One and All depending whether a single line item conflict or a category is selected from the error log. An error report is generated listing the x-y position and the elevation difference of the entities in conflict.

Draw All/One toggles between One and All depending whether a single conflict or a category is selected from the list. This option draws an "X" symbol at each selected conflict. The layer and size of the symbol is controlled in the fields below.

Continue closes the Error Log and proceeds with the contouring operation.

Settings has controls for the tolerances for error reporting and for the Layer Name and Symbol Size to use with the Draw function.

Keyboard Command: mk_exist_tin
Prerequisite: a boundary polyline and elevated entities on the Existing layer target

Make Design Surface

This command makes the triangulation models for the design surface. There are three surfaces that are created: initial design, design after applying subgrade zones, and design after subgrade zones and topsoil replacement. These surface files are automatically named as "filename-bs.tin", "filename-zn.tin" and "filename-fn.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also, the file extension will be .tin for the binary format triangulation and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Design surface, then the design after subgrades surface with be the same as the initial design surface. The topsoil replacement depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil replacement areas will lower the design surface by the topsoil depth to leave room for the topsoil replacement. If there aren't any topsoil replacement areas, then the design after subgrade and topsoil surface will be the same as the design after subgrade surface.

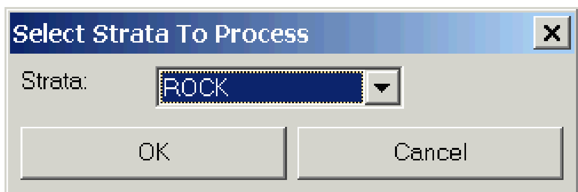
Before running this command, the layer names for the entities on the Design layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site.

When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Refer to the Make Existing Surface command for more information on the Data Error Log dialog.

Keyboard Command: mk_final_tin
Prerequisite: a boundary polyline and elevated entities on the Existing layer target

Make Overexcavate Surface From Strata

This command sets the Overexcavate surface to a selected strata surface. Before running this command, the strata surface must be created with the Make Strata Surfaces command in the Drillhole menu. The resulting overexcavate surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.



Prerequisite: Strata surfaces
Keyboard Command: overx_by_strata

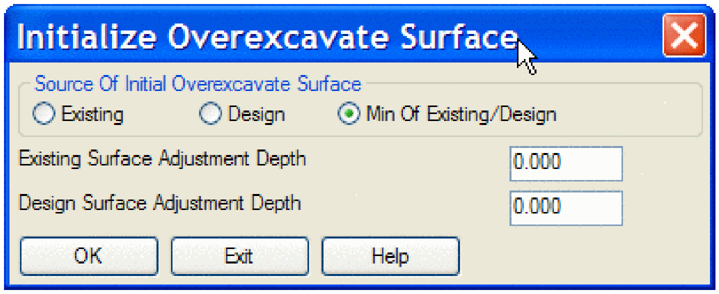
Make Overexcavate Surface From Screen Entities

This command makes the overexcavate surface from entities on the layers defined as Overexcavate in the Define Layer Target/Material/Subgrade command. The resulting surface of Make Overexcavate Surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.

Prerequisite: overexcavate entities
Keyboard Command: mk_overx_tin

Make Overexcavate Surface From Existing/Design Surfaces

The Initialize Overexcavation Surfaced dialog box shown here allows the user to select which surface model to overexcavate and to enter in the depth value for the desired adjustment. Use the Min Existing/Design option to set the overexcavate as the minimum of the existing and design surfaces. If a single surface is selected the value entered will be applied to that surface only. The resulting surface of Make Overexcavate Surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.



Prerequisite: Existing and/or Design surfaces
Keyboard Command: set_overx

Adjust Overexcavate Surface

This command adjusts the overexcavate surface vertically within the selected perimeter polylines. This command allows the site to be overexcavated at a variety of depths in specified areas represented with CLOSED polyline boundaries. Select the desired areas to be adjusted when prompted at the command line.

Keyboard Command: adjust_overx
Prerequisite: an overexcavate surface

View Overexcavate Surface

Use this command to view the current overexcavate surface. The Takeoff 3D Viewer will display the 3D faces of the adjusted surface. Shade the 3D model and adjust its perspective to view a rendered display. The surface that is displayed will depend on the latest surface created using the make and adjust routines.

Prerequisite: an overexcavate surface
Keyboard Command: cube_overx

Draw Overexcavate Surface 3D Faces

Use this command to draw the 3D faces of the overexcavated surface model on the screen. The 3D faces will be drawn in the TK_OVERX_SURFACE layer and will depend on the latest surface created using the make and adjust

routines.

Prerequisite: An overexcavate surface
Keyboard Command: draw_overx

Erase Overexcavate Surface 3D Faces

Use this command to remove the previously drawn 3D Faces from the screen.

Prerequisite: 3D Faces
Keyboard Command: erase_overx

Draw Overexcavate Cut Color Map

Use this command to display a cut color map on the screen that shows the areas of overexcavate cut. The colors will graduate from white to red based on zero cut depth to maximum cut depth. This command also offers the user to place a legend of the cut depths on the screen. Pick the desired location and type the desired scale of the legend when prompted at the command line.

Prerequisite: An overexcavate surface
Keyboard Command: overx_cfmap

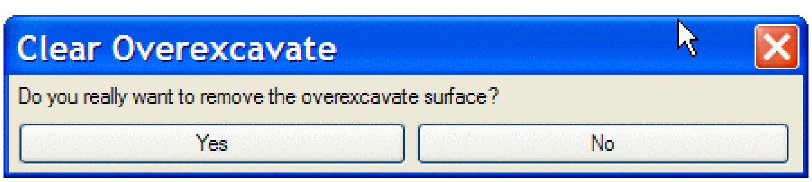
Erase Overexcavate Cut Color Map

Use this command to remove the previously drawn Cut Color Map and Legend from the screen.

Prerequisite: An overexcavate cut color map
Keyboard Command: overx_cfmap2

Clear Overexcavate Surface

Use this command to remove the overexcavate surface. When the overexcavate surface is removed, the rest of the Takeoff commands will not calculate overexcavate volumes. You will be prompted to confirm before the remove is done.

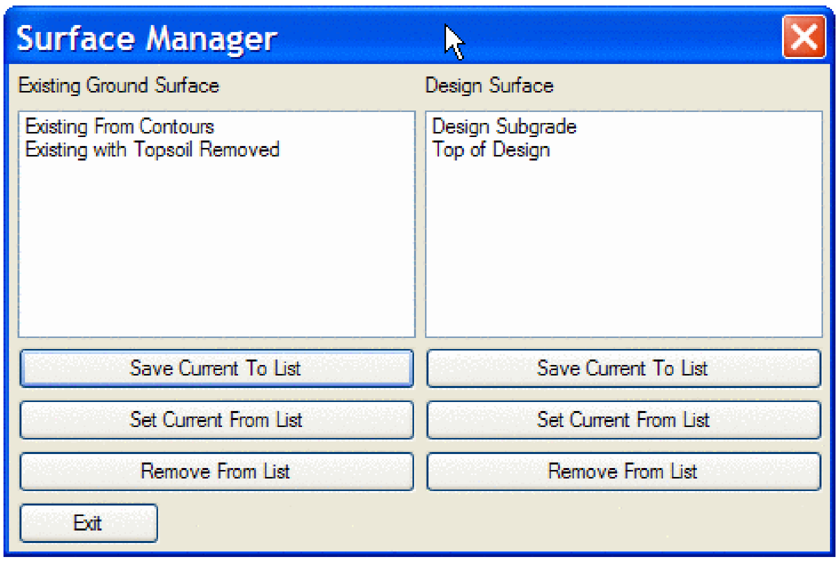


Prerequisite: An overexcavate surface

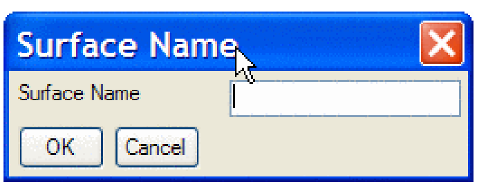
Keyboard Command: clear_overx

Surface Manager

This command allows the user to name and manage multiple surface models. The Surface Manager dialog shown here has options to name and save the current "existing and design" surface models. The "current" surface is dictated by the layers that populate a target and the Make Surface command. If layers are removed from a target, and others assigned, multiple surfaces can be created and stored. Highlight a named surface and select the Set Current From List option to make that model active. Use the Remove From List option to remove a named surface model from the list.



Selecting the Save Current To List options brings up the Surface Name dialog box shown here. Type the desired name that describes a particular surface model and select OK.



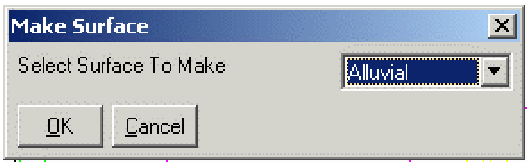
Prerequisite: none
Keyboard Command: surf_mgr

Make User Defined Surface

This command makes a surface from the entities on the layers defined as user-defined targets in the Define Layer Target/Material/Subgrade command. The purpose of user-defined surfaces is for modeling surfaces besides existing ground and design. The drawing needs to contain entities that represent the elevations of the user-defined surface. For example, the user-defined surface could be for alluvial soil and the drawing has contour polylines for this surface.

There is a dialog to select which surface to make. The surface is stored in a triangulation file that is named after the current drawing name with the user-defined surface name appended.

This user-defined surface can be applied to Takeoff routines by running the Set Active Surfaces command.

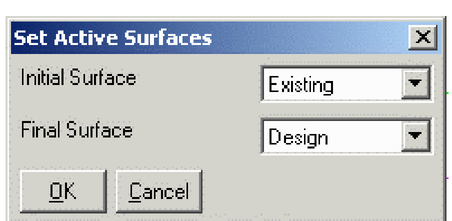


Prerequisite: Define Layer Target/Material/Subgrade command
Keyboard Command: mk_user_tin

Set Active Surfaces

This command assigns which surfaces to use for initial and final. These surfaces are used by all the Takeoff routine that compare surfaces including:
- Calculate Total Volumes
- Calculate Volumes Inside Perimeter
- 3D Drive Simulation
- Cut/Fill Contours/Labels/Color Map
- Surface Inspector
- Quick Profile
- etc.

The surface created by the Make Existing Ground Surface command is called "Existing" and is the default for the Initial Surface. The surface created by the Make Design Surface command is called "Design" and is the default for the Final Surface.



The purpose of this routine to for selecting user-defined surfaces to use in place of the existing ground or the design surface. For example, there could be a user-defined surface for alluvial soil that is set as the initial surface while design is set to the final surface. Then the calculate volume routines will report the quantities between alluvial soil and design. Also the Display->Cut/Fill Color Map routine will make the map for the difference between the alluvial soil and design surfaces.

These user-defined surfaces can be created using the Add Target function in the Define Layer Target/Material/Subgrade command combined with the Make User-Defined Surface command.

Prerequisite: a surface model
Keyboard Command: set_active_tins

Design Surface Vertical Offset

This command can be used to lower or raise the design surface within a defined perimeter or by the entire surface.

Prerequisite: a design surface
Keyboard Command: adjust_final

Existing Surface Vertical Offset

This command can be used to lower or raise the existing surface within a defined perimeter or by the entire surface.

Prerequisite: an existing surface
Keyboard Command: adjust_exist

Merge Existing With Design

This command allows you to merge the existing surface with design surface within perimeter polylines. The resulting merged surface can be saved to update either the Existing or Design surfaces. The program prompts for inclusion and exclusion perimeter polylines. These polylines must be closed. The merge will be applied inside the inclusion perimeters and not inside the exclusion perimeters. The exclusion perimeters are optional.

For example, if a portion of the site is completed, you can update the existing surface to match the design for the completed area. First, draw a closed polyline around the completed area. Then run Merge Existing With Design and choose the merge results target as Existing. Then select the perimeter polyline.

Prerequisite: existing and design surfaces and an inclusion perimeter polyline
Keyboard Command: merge_final

Triangulate & Contour

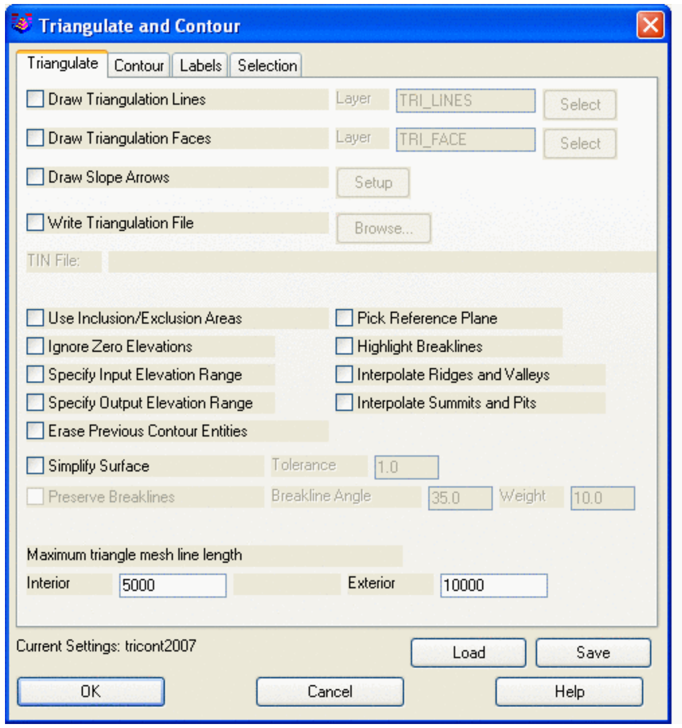
This command provides a complete set of functionality for contouring, labeling, and creating tin surface models. Given data entities that represent the surface, this command creates a final contour map with labeled, smoothed, and highlighted contours and/or a surface model that can be saved to a file (to be used in other areas of the program) or drawn on the screen as triangles or faces. Eligible data entities include points, inserts, lines, 2d polylines, 3d polylines, elevation text, 3d faces, and points from ASCII or coordinate (.CRD) files.

Triangulate & Contour has many options which are defined in the exhibits shown in the following pages. With this command, you can do any combination of drawing the triangulation network lines, drawing the contours, drawing triangulation network 3D Faces or lines, writing a triangulation file and storing a surface file.

In order to force *Triangulate & Contour* to correctly interpolate elevations between two points that define a grade break in the surface (such as points on a ridge, wall, or road), a breakline must exist between the points. A breakline line can be specified as a 3D polyline or line. In fact, all 3d polylines and lines with elevation are treated as breaklines.

If *Triangulate & Contour* reports zero points found and fails to do anything when you're using Carlson points, then those points are probably located at zero elevation. To fix this problem, make sure that Carlson Point Inserts is toggled on in the Selection tab. This will enable *Triangulate & Contour* to read the elevation from the elevation attribute of the point.

Triangulate Tab



When **Draw Triangulation Lines** is turned on, the program will draw the triangulation as simple AutoCAD lines with elevation. Specify the layer for these lines in the box to the right.

When **Draw Triangulation Faces** is turned on, the program will draw each triangle in the triangulation network as a 3D Face. These 3D Faces can then be used in AutoCAD's modeling routines such as *HIDE* and *SHADE* or in Carlson routines such as *3D Viewer Window, 3D Surface FlyOver and Slope Zone Analysis*. Specify the layer for these 3DFaces in the box to the right.

Write Triangulation File stores the triangulation surface model as an .flt or a .tin file. The .flt file format is a text file depicting the edges in the triangulation network. The .tin file is a new binary file format depicting the triangulation network. The .tin file is much faster and more efficient than the previous .flt file format. The triangulation file(s) can be used by several commands such as *Volumes By Triangulation*, *Spot Elevations, and Profile from FLT File*. Either type in the file name to create or press the Browse button to select a file name.

When **Use Inclusion/Exclusion Areas** is activated, the program will prompt you for inclusion and exclusion polylines. These are used to define the area of activity for triangulation and contouring. The inclusion and exclusion polylines must be closed polylines and must be drawn before using *Triangulate & Contour*. The command line display must be set to show at least two lines to see the prompting for the selection of the Inclusion/Exclusion perimeters.

Only the parts of the contour lines and triangles that are within the inclusion polylines will be drawn. For example, an inclusion could be the perimeter of the site. The parts of contour lines that are inside the exclusion polylines are not drawn. Exclusion polylines can be used for areas where you don't want contours such as within buildings. When **Ignore Zero Elevations** is activated, this setting will filter out all data points at an elevation of zero from the data set.

If you would like to manually set the range over which to contour, select one of the **Specify Input/Output Elevation Range** options, one for source data and one for contour output. The program will automatically contour from the lowest elevation in the data set up to the highest at the increment specified in Contour Interval.

When **Erase Previous Contour Entities** is activated, this setting will erase previously drawn contour entities.

The triangulation network is based on the x,y position of the points. **Pick Reference Plane** allows you to contour an

overhang or cliff by changing the reference plane to a side view. The reference plane can be specified by first using the *Viewpoint 3D* command and then using the View option, or you can specify three data points on the cliff (two along the bottom and one at the top).

Highlight Breaklines highlights breaklines in the triangulation network by drawing the triangulation lines along breaklines in yellow.

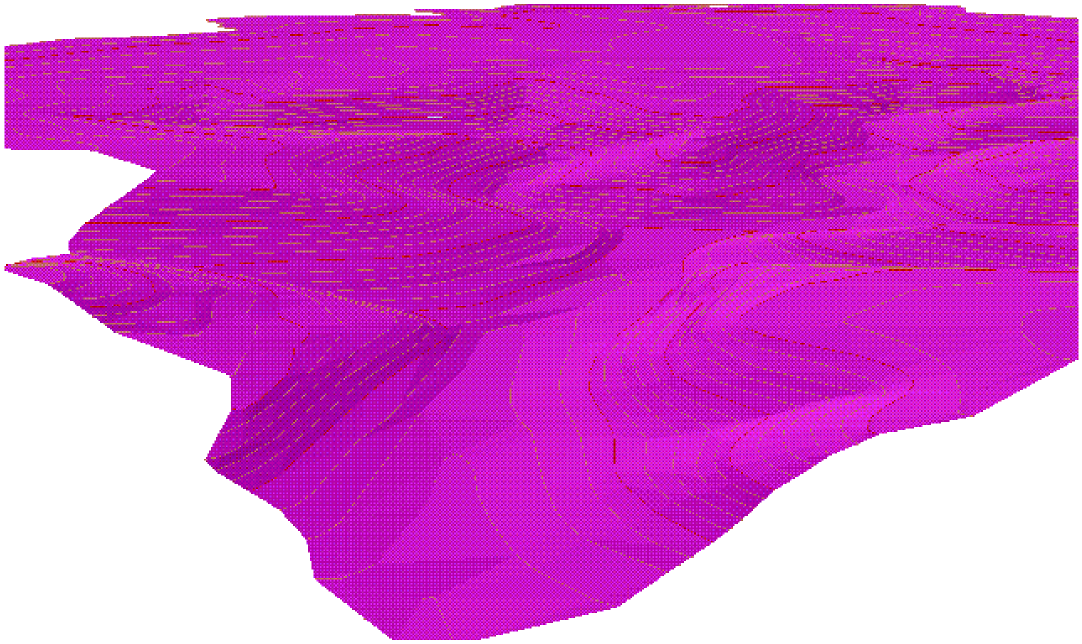
Interpolate Ridges and Valleys creates additional triangulation in a ridge or valley situation to more accurately define the feature during surface modeling operations. This option would commonly be used when creating a surface model from existing contours, since it replaces the need to manually draw 3d polylines along ridges and valleys.

Interpolate Summits and Pits creates additional triangulation in a summit or pit situation to more accurately define the feature during surface modeling operations. This option would commonly be used when creating a surface model from existing contours.

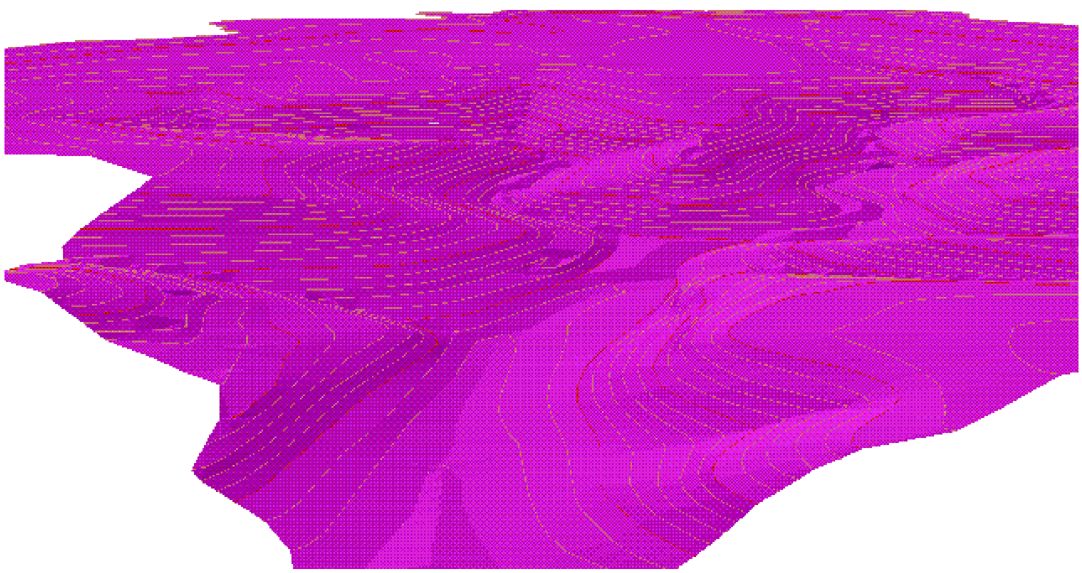
Simplify Surface is designed to reduce the digital size of a surface, without significantly compromising the integrity or accuracy of the surface. The most common application is for surfaces derived from very large datasets, such as smoothed contours. Its use is less applicable to design surfaces or surfaces based on surveyed points, but it can still be utilized. **Simplify Surface** reduces the size of the surface file by analyzing the difference in elevation between each vertex of the TIN and the vertices directly surrounding it, assigning a numerical weight or value to each vertex. If it is determined that the calculated weight for a particular vertex is less than the **Tolerance** factor, the vertex is a candidate for removal. The number of vertices removed is directly proportional to the **Tolerance** factor, so the higher the **Tolerance** factor, the more vertices are removed, the lower the **Tolerance** factor, the fewer vertices are removed.

If **Simplify Surface** is selected, the **Preserve Breaklines** option is activated. **Preserve Breaklines** further analyzes the TIN by focusing on the edges, calculating the angular difference between adjacent triangular faces. If the angular difference between edges is greater than the specified **Breakline Angle**, it is considered to be a breakline, and it is preserved. If it's angular difference is determined to be below the **Breakline Angle**, it becomes a candidate for removal. In that case, the **Weight** factor is applied to the corresponding vertex, adjusting it's original value. If the resulting value is still below the **Tolerance**, it is then removed. The number of vertices removed is inversely proportional to the **Weight** factor**,** sothe greater the **Weight** factor, the fewer vertices that are removed, the lower the **Weight** factor, the more vertices that are removed.

A good rule of thumb that can be used when deciding whether or not to use these options is: if the surface contains no man-made features, use **Simplify Surface** only, if it contains man-made features, such as roads, use both **Simplify Surface** and **Preserve Breaklines**.



Before: Surface made from an existing contour map. Note the flat spots in the bottom of the valley (bottom center of the image) when Interpolate Ridges and Valleys is disabled.



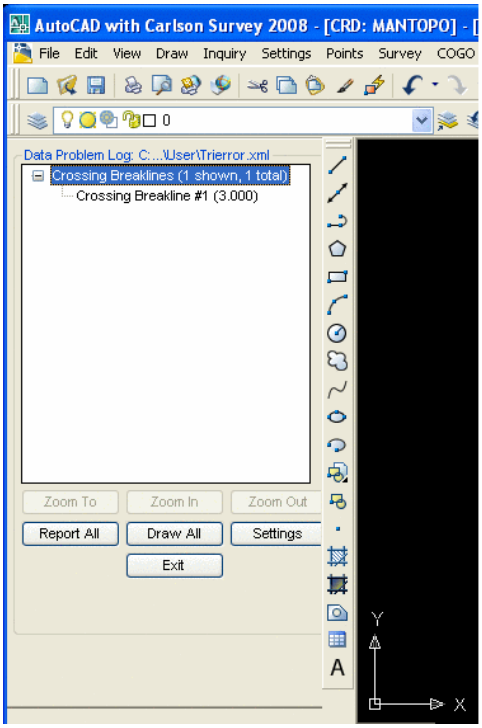
After: The same surface with Interpolate Ridges and Valleys enabled. Note the smooth flowline at the bottom center of the image.

The **Max Triangle Mesh Line Length** value limits the length of the triangulation network lines. Any triangulation line that exceeds this limit will not be drawn or included in contouring. This allows you to avoid abnormally long triangulation lines where you have relatively too few data points and on the outskirts of your data points. The **Exterior** value applies to triangulation lines around the perimeter of the triangulation area and the **Interior** value applies all the other triangulation lines. Generally you would have the exterior value larger than the interior.

An **Error Log** is generated if the *Triangulate and Contour* routine finds a vertical conflict between breaklines or other surface entities, opening the following dialog box. Three types of conflicts are reported; Crossing Breaklines, Vertical Edges, and Breakline T-Intersections. Crossing Breaklines indicates that the intersection of two entities does not have a common elevation. Vertical Edges indicates that two entities or vertexes of differing elevations have

the same x-y location, thus forming a vertical plane. Breakline T-Intersections indicates that a 3d entity is abutting another entity, but the second entity doesn't have a vertex at the point of intersection. Each type of conflict is listed in its own category.

Clicking to the "plus" sign beside a category will display the individual conflicts within that category. When a line item error is selected, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Zoom functionality allows the user to more closely inspect the specific problem area, and if needed a marker can be drawn or a report generated for an individual conflict or conflicts.

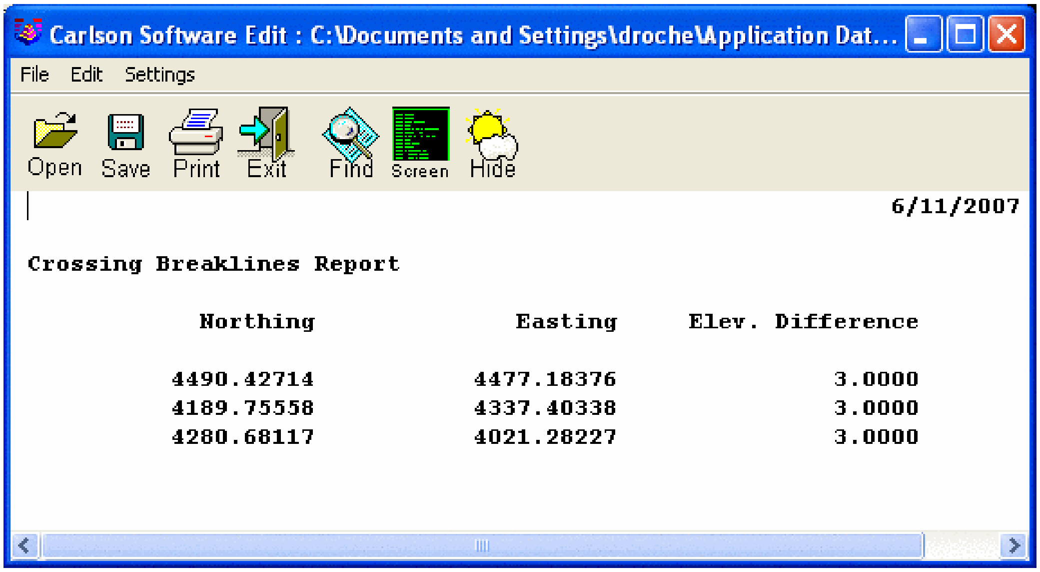


Zoom pans the drawing to move the selected conflict to the center of the screen. The zoom functions are only active when a single line item is selected.

Zoom In zooms in on the highlighted area for closer inspection**.** Multiple picks on the zoom button will increase the magnification.

Zoom Out zooms out away from the highlighted area.

Report All/One toggles between One and All depending whether a single line item conflict or a category is selected from the error log. An error report is generated listing the x-y position and the elevation difference of the entities in conflict.



Draw All/One toggles between One and All depending whether a single conflict or a category is selected from the list. This option draws an "X" symbol at each selected conflict. The layer and size of the symbol is controlled in the fields below.

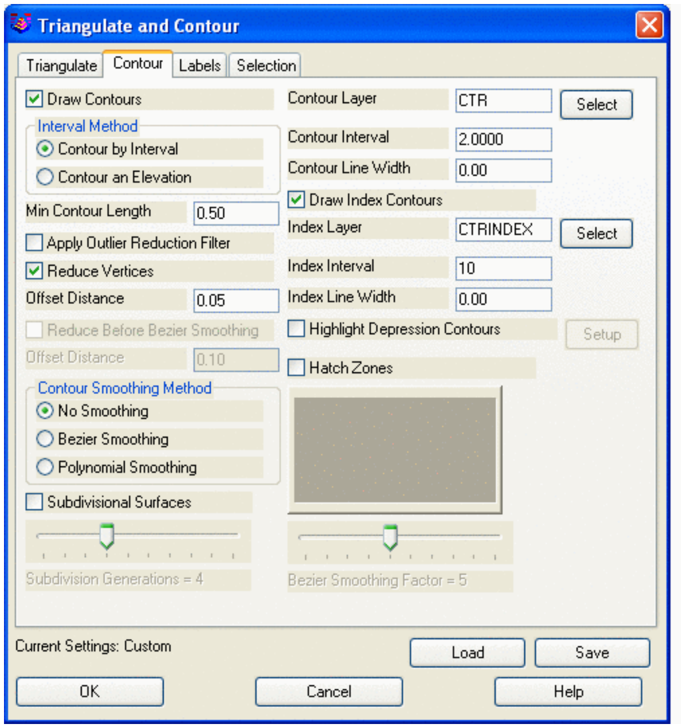
Continue closes the Error Log and proceeds with the contouring operation.

Layer Name specifies the layer name for the "X" entities drawn with Draw One/All. This also sets the layer name for the "Draw Lines" option.

Symbol Size specifies the size of the "X" symbol that is drawn to delineate the selected errors. This will determine the actual size of the symbol in the drawing. This value is not multiplied by the horizontal drawing scale.

In the case of crossing polylines, **Draw Lines** will trace over the polylines responsible for the conflict. The polylines will be created in the layer specified in the layer field.

Contour Tab



When the **Draw Contours** box is checked, the program will draw contour lines after triangulating. Otherwise, only the designated triangulation operations are performed. Specify the layer for contours in the edit box to the right.

Contour by Interval or **Contour an Elevation** determines whether to contour by interval (ie: every 10 feet) or to contour a certain elevation. The elevation option allows you to contour specific values. For example, if you want just the 100ft contour, then select elevation and enter 100. The default mode is by interval.

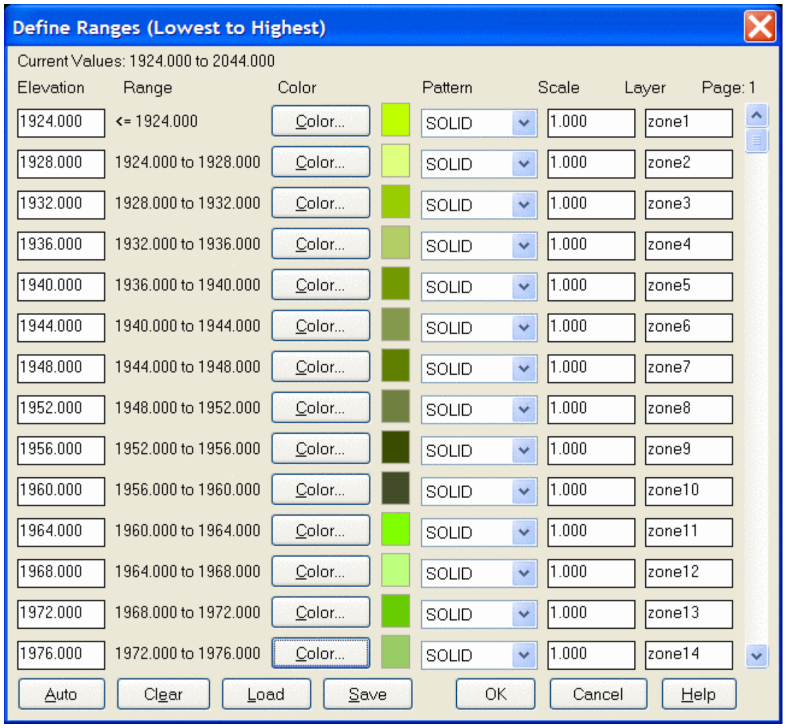
Use **Contour Interval** to specify the interval to contour. **Note:** If the previous option is set to Contour an Elevation, then this field is used to specify the elevation to contour.

Contour lines whose total length is less than the **Min Contour Length** value will not be drawn.

Reduce Vertices attempts to remove extra vertices from the contour polylines which has the advantages of a faster drawing and smaller drawing size. Default is ON

When the Reduce Vertices option is enabled, the **Offset Distance** value is the maximum tolerance for shifting the original contour line in order to reduce vertices. The reduced contour polyline will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour line to shift more from the original.

When activated, the **Hatch Zones** option will create hatching between the contours based on elevation zones. The following dialog will open allowing the user to specify the hatch type and color for each elevation zone. The entire elevation range of selected data is displayed under Current Values.

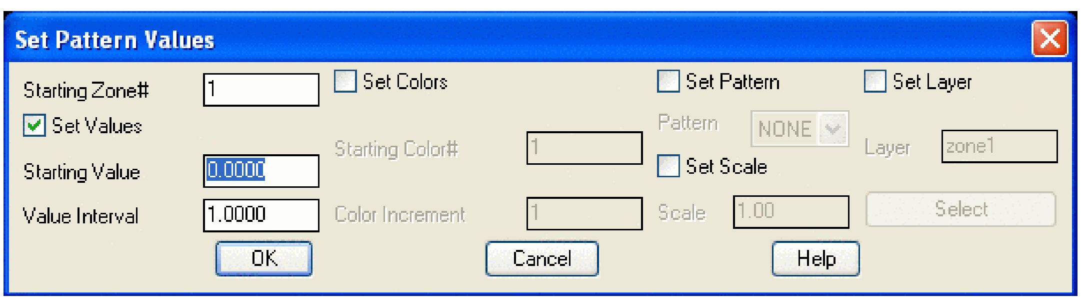


Clear clears the all of the Elevation fields in the dialog.

Load loads previous settings from a saved .pat file.

Save saves the current setting configuration to a .pat file.

Auto opens the following dialog, allowing for automatic configuration of the range of elevations in each zone, assigning of colors and hatch patterns, and the scale.



Starting Zone sets the zone with which to begin the application of the setting defined in this dialog. For Instance, if the Starting Zone was set to 10, the settings definitions applied here wouldn't affect Zones 1-9, but would start at Zone 10.

Set Values enables the Starting Value and Value Interval fields, which allow the user to specify the starting elevation for the given zone and set the zone increment.

Starting Value sets the elevation of the beginning zone to define.

Value Interval sets the elevation increment for subsequent zones.

Set Colors enables the Starting Color and Color Increment fields.

Starting Color sets the starting color number, based on the AutoCAD standard color chart.

Color Increment sets the color number to increase for subsequent zones. So if the increment was set to 5, and the starting color was 60, the next color would be 65, 70, and so on.

Set Pattern sets the hatch pattern for the defined zones.

Set Scale enables the Scale option.

Scale sets the scale for the selected hatch pattern.

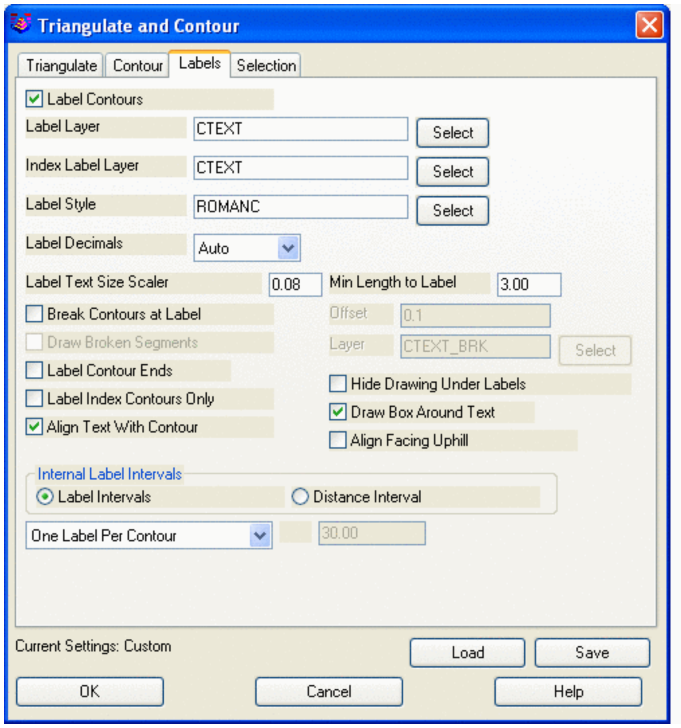
Draw Index Contours creates highlighted contours at a specified interval. When enabled, the fields for Index Layer, Index Interval and Index Line Width are activated.

Use **Contour Smoothing Method** to select the type of contour smoothing to be performed. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces. The smoothing factor described below affects the smoothing bulge.

Bezier Smoothing Factor The contour preview window shows you an example of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect. Note that too much smoothing applied in some situations can result in crossing contours.

Subdivisional Surfaces / Subdivisions Generation causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation, with the new temporary vertices raised or lowered to provide smoother contours. More generations increases the smoothness of the algorithm at a cost of increased processing time. If Straight Lines are chosen as the contouring drawing method, then the contours are guaranteed never to cross. The original points of the surface model are always preserved. These modifications to the surface model are only for contouring purposes and are not written to the triangulation (.FLT) file or inserted into the drawing. If some contour movement is too small for appearance's sake, consider enabling Reduce Vertices.

Labels Tab



When **Label Contours** is activated, contours will be labeled based on the settings below.

Label Layer specifies layer name for intermediate contour labels.

Index Label Layer specifies layer name for index contour labels.

Label Style specifies the text style that will be used for the contour label text.

Label Text Size Scaler specifies the size of the contour labels based on a multiplier of the horizontal scale.

Contours whose length is less than the **Min Length to Label** value will not be labeled.

When **Break Contours at Label** is checked, contour lines will be broken and trimmed at the label location for label visibility. When enabled, the Offset box to the right activates. The Offset determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

When **Draw Broken Segments** is checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

When **Label Contour Ends** is checked, contour ends will be labeled.

When **Draw Box Around Text** is checked, a rectangle will be drawn around contour elevation labels.

When **Label Index Contours Only** is checked, only the index contours will be labeled. This option is active only when "Draw Index Contours" has been selected in the Contour tab of the main dialog.

Hide Drawing Under Labels activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour line is still fully intact. This feature provides the user with

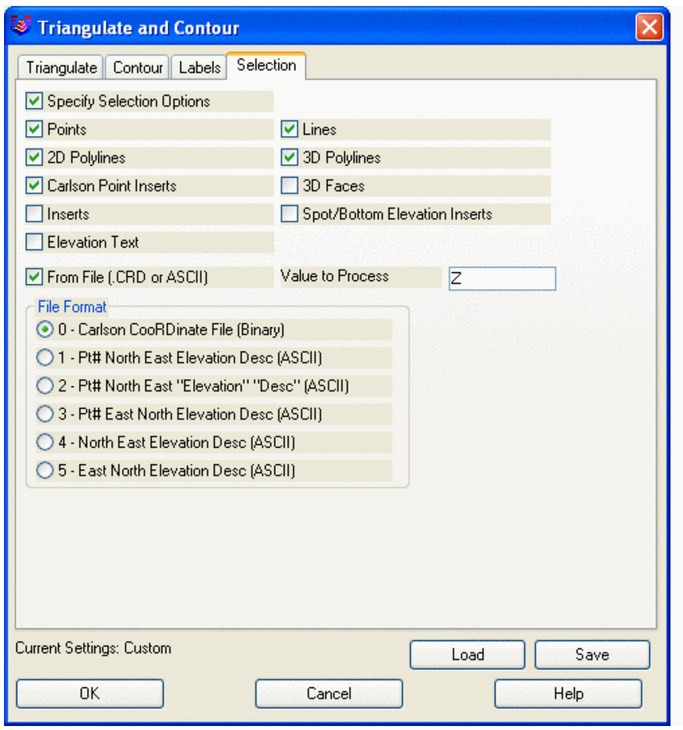
the best of both worlds; you have clean looking contour labels, yet the contour lines themselves remain contiguous. This feature will also hide other entities that are in the immediate vicinity of the contour label.

When **Align Text with Contour** is checked, contour elevation labels will be rotated to align with their respective contour lines. This option also activates the Align Facing Uphill feature explained below.

When **Align Facing Uphill** is checked, contour elevation labels will still be rotated to align with their respective contour lines, but the labels will be flipped in such a manner that the top of the text label will always be toward the uphill side of the contours. So as the labels are read right side up, the contours will be progressing uphill.

Use **Internal Label Intervals** to choose between Label Intervals or Distance Interval. Label Intervals will label each contour with a set number of labels. Distance Interval lets you specify a distance between labels.

Selection Tab



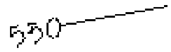
When **Specify Selection Options** is checked, you can control what type of entities *Triangulate & Contour* uses. This is an excellent method of "filtering out" unwanted entity types.

Points, 3D Polylines, 2D Polylines, Lines, Inserts are standard AutoCAD entities types.

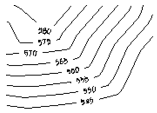
Carlson Point Inserts refer to Carlson points which include the block SRVPNO∗ with the point number, elevation, and description attributes.

Spot/Bottom Elevation Inserts include text entities that start with 'X'.

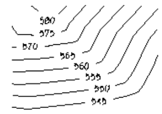
From File allows you to triangulate from the points in a coordinate (.CRD) or ASCII file. This option also provides access to the use of Point Groups as a data source.



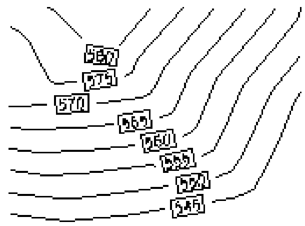
Label Contour Ends



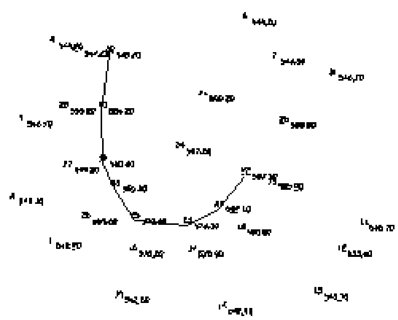
Align Text With Contour ON



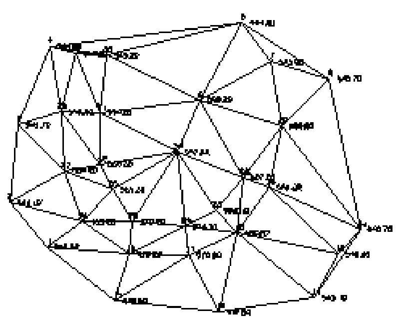
Align Text With Contour OFF



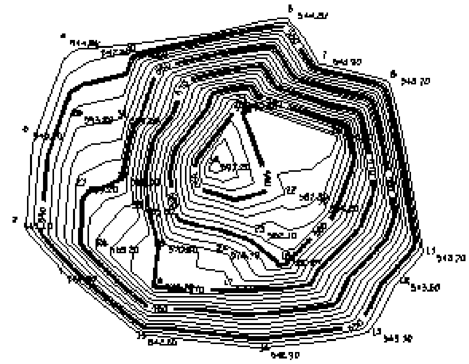
Draw Box Around Text



Original data points with one 3D polyline



Triangulation network without contouring



Contours without triangulation network
The contours are smoothed, reduced, drawn at an interval of 2, and highlighted at an interval of 10 with labeling on the index contours.

Pulldown Menu Location: Surface
Keyboard Command: tri
Prerequisite: Data entities in the drawing, including points, inserts, lines, 2d polylines, 3d polylines, elevation text, 3d faces, and points from ASCII or coordinate (.CRD) files.
File Names: \lsp\tri4.lsp, \lsp\tri4.dcl, \lsp\tri4.arx

Triangulation File Utilities

This command allows you to modify TIN surfaces in a variety of different ways, then allows for 3d viewing and shading of the modified surface and finally for saving the file with a choice of output formats. The focus of the routine is to elevate or lower the TIN or selected areas within the TIN, merge TINs with other surfaces, or use data from other TIN files to apply to the current TIN. Operations can be performed on the entire TIN or just on user selected Inclusion and/or Exclusion areas. The routine will automatically rework the TIN network for conformation to a selected boundary, say a building outline. In the case of said building, a value of 10 could be subtracted from the building outline. This will drop all of the triangulation within the outline by 10', thus creating a model of the excavated area for the building. The modified TIN can then be saved to a new file, which could be used to compute an excavation volume with Volumes by Triangulation. This routine does not allow for manual reconfiguration of the TIN network. This is performed under Surface Tools, also in the Contour pulldown menu. This routine also includes conversions to and from TIN files, DXF files and 3D Face entities.

Begin with the dialog shown here. First select a TIN model. You may choose between an .flt or .tin file, a DXF file (that includes 3DFACE entities), or 3DFACE entities in the current drawing. Specify the subject area by choosing inclusion or exclusion perimeters, then press the next button.

Load TIN File: Allows you to specify a triangulation (.flt or .tin) file to load.

Load DXF File: Allows you to specify a DXF file to load. Only loads 3DFACE entities from the selected DXF file.

Select 3D Faces: Allows you to select 3DFACE entities from the current drawing. This also includes rectangular 3d faces from a plotted grid.

Pick Bounding Polylines: Allows you to select any inclusion/exclusion perimeter(s). When this button is selected, the user is taken back to the drawing and prompted to select the perimeters. Press Enter when the selections are finished to return back to the dialog.

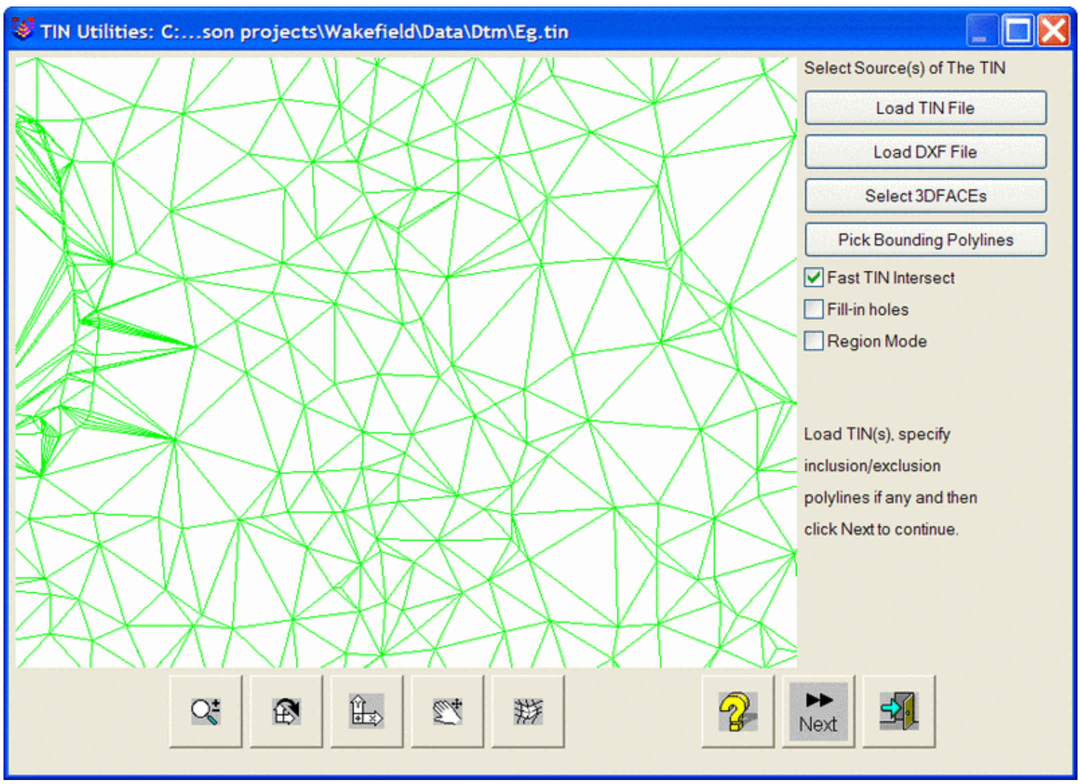
Fast TIN Intersect: When checked, this command will perform a simple and fast check for overlapping triangles, so is the preferred choice in most cases. However, if problems with the TIN are suspected, this option should be unchecked, so that a complete and thorough check and repair of the TIN is performed.

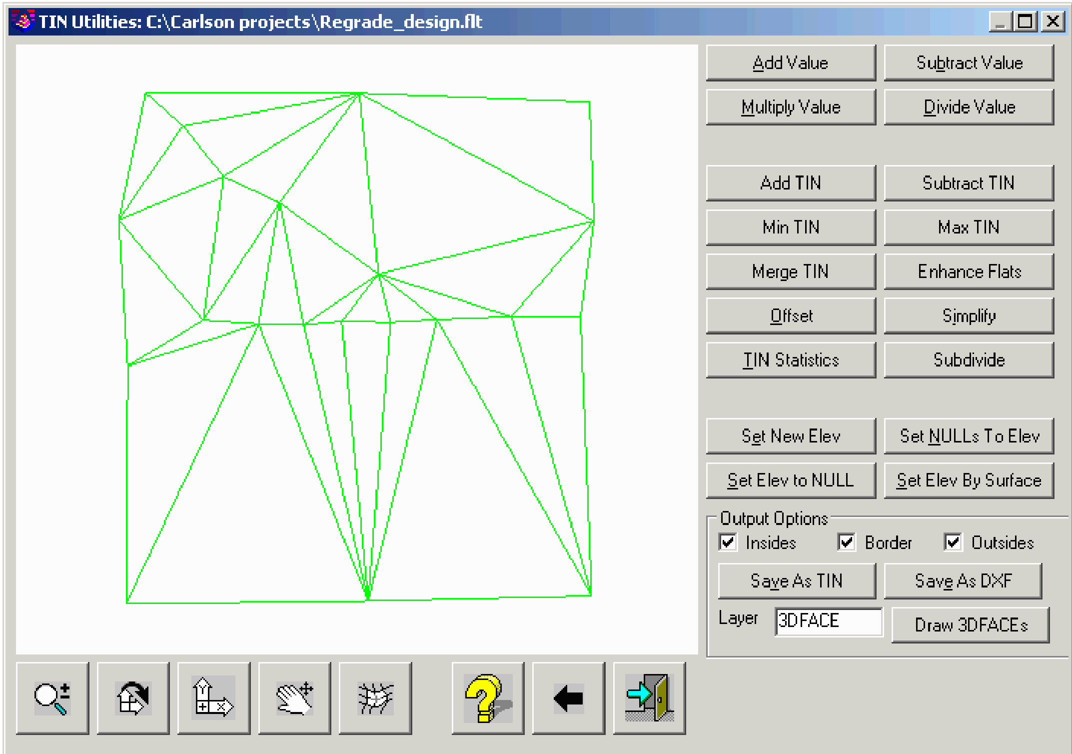
Fill-in-holes: When checked, any missing triangulation or gap in the surface will be automatically filled in with additional triangles. This option has to be set before loading the TIN file to take effect.

Region Mode: This option deals with nested or overlapping boundaries. When checked, AutoCAD hatch pattern logic is applied, in which all nested boundaries are used in an alternating fashion, so that an Inclusion Boundary within an Exclusion Boundary is still recognized. If this option is not checked, everything within an Exclusion Boundary is ignored.

Next: Press this button to proceed to the next dialog after all selections have been made .

The next dialog allows you to perform mathematical operation(s) on the loaded TIN. Each operation is described below. Keep in mind that generally these operations are to be performed on an area inside your inclusion perimeter (but excluding anything inside your exclusion perimeters). If you do not specify any perimeters, the desired operation/s will be performed on the entire TIN.





Add Value: Prompts for a value to Add to the subject area of the TIN**.**

Subtract Value: Prompts for a value to Subtract from the subject area of the TIN**.**

Multiply Value: Prompts for a value to Multiply to the subject area of the TIN**.**

Divide Value: Prompts for a value to Divide to the subject area of the TIN**.**

Add TIN: Raises the subject area of the current TIN by the elevation value from a second user selected TIN file. This function is most applicable to applying a strata thickness TIN.

Subtract TIN: Lowers the subject area of the current TIN by the elevation value from a second user selected TIN file.

Min TIN: This does a comparison between the current TIN and a second user selected TIN file, and applies the lower value of the two TINs to the subject area.

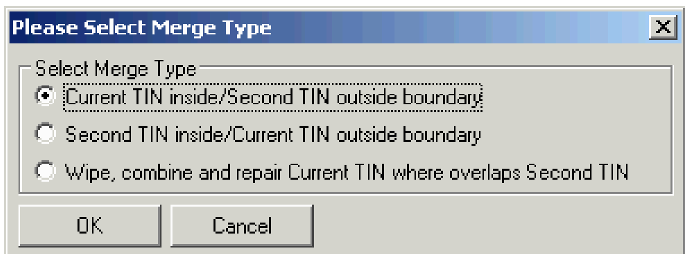
Max TIN: This does a comparison between the current TIN and a second user selected TIN file, and applies the higher value of the two TINs to the subject area.

Merge TIN: Merges the current subject TIN into a second user-specified TIN file. There are three methods:
Current TIN inside/Second TIN outside boundary: This method is only available when Bounding Polylines are selected in the first Triangulation File Utilities dialog. The current TIN will be used inside the boundary polylines and the second TIN is used everywhere else. The current TIN file should be the smaller of the two surfaces since the

subject file will be joined or merged into the second file. For example, to merge a pad design into existing ground with this method, choose the pad design as the current TIN, pick the pad perimeter as the bounding polyline and use existing ground as the second TIN.

Second TIN inside/Current TIN outside boundary: This method uses the second TIN inside the boundary and the current TIN everywhere else. The outline of the second TIN is used as the boundary if no bounding polylines where selected in the initial dialog. For example, to merge a pad design into existing ground with this method, choose the existing ground as the current TIN and choose the pad design as the second TIN.

Wipe, combine and repair Current TIN where overlaps Second TIN: This method removes triangles from the current TIN for areas that overlap the second TIN. Then the second TIN is added into the current TIN surface and the gap between the current and second TINs is triangulated to stitch them together. This method is useful when the two TINs don't have matching have elevations on their common boundary. Then this method will create a transition zone between the TINs.



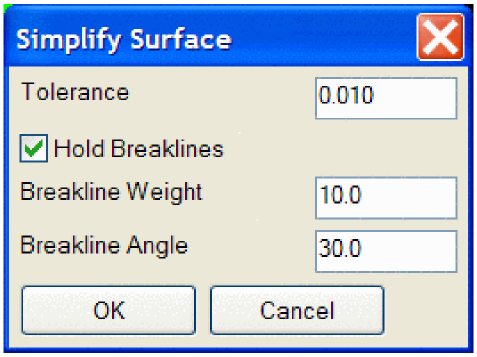
Enhance Flats: This routine eliminates flat triangles by adding a data point inside the triangle at a different elevation to subdivide the triangle. The elevation of this point is calculated based on the slopes of the neighboring triangles.

Offset: Performs a perpendicular offset (from the face/s) to the TIN surface by the specified amount.

Simplify: Causes edges within the Tin mesh to be collapsed to reduce the number of triangles, edges, and points within the mesh while having a minimal impact on the overall shape of the mesh.

Tolerance: This setting is used by the Simplify command described below. Specify the maximum average distance that any point can be moved outside of the plane of any triangle that connects to that point. Values might range from .01 to .1 for most purposes.

Hold Breaklines: Further analyzes the TIN by focusing on the edges, calculating the angular difference between adjacent triangular faces. If the angular difference between edges is greater than the specified **Breakline Angle**, it is considered to be a breakline, and it is preserved. If it's angular difference is determined to be below the **Breakline Angle**, it becomes a candidate for removal. In that case, the **Breakline Weight** factor is applied to the corresponding vertex, adjusting it's original value. If the resulting value is still below the **Tolerance**, it is then removed. The number of vertices removed is inversely proportional to the **Breakline Weight** factor, sothe greater the **Breakline Weight** factor, the fewer vertices that are removed, the lower the **Breakline Weight** factor, the more vertices that are removed.



TIN Statistics: Generates a report of the TIN statistics, including number of points, edges, and triangles, and minimum and maximum Z value.

Subdivide: Subdivides triangles to make them more equalateral.

Set New Elev: Sets all TIN faces in the subject area to the elevation specified.

Set NULLs to Elev: Sets all NULL values in the subject area to the elevation specified.

Set Elev to NULL: Sets all of the elevation values in the subject area to NULL.

Set Elev by Surface: Sets all TIN faces within the subject area to the elevations from a second surface file within the same area. You will be prompted to select a second TIN file or grid file. Only areas common to both surfaces will be applied to the subject TIN.

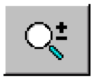
Output Options: The following three options determine what part or parts of the TIN modifications that will be saved to the new TIN file. If the entire TIN is to be saved, all three options should be toggled on.

- **Insides:** If this is the only option checked, only changes made to the TIN within the inclusion perimeter will be saved. TIN entities outside of the perimeter will not be saved to the named file.
- **Border:** When the routine re-works the TIN to fit around a perimeter, a small horizontal offset is automatically applied to prevent the formation of vertical faces. The Border function will save changes made to TIN in this offset area.
- **Outsides:** If this is the only option checked, TIN entities inside of the inclusion perimeter will not be saved to the named file. Everything outside of the perimeter will be saved.

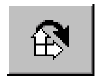
Save As TIN: Saves the current TIN as an .flt or .tin file.

Save As DXF: Saves the current TIN as a .dxf file. This format can be used by many other CAD programs.

Draw As 3DFaces: Draws the current TIN as 3D Faces in the current viewport. The Layer window is used to specify the layer that the faces will be created in.

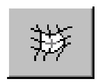


Converts the left mouse button to a zoom function. Hold the button down and move the mouse up or down to zoom in and out.

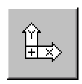


Converts the left mouse button to a rotate function. Hold the button down to rotate the view in any X, Y or Z direction. When the XY appears in the window, the rotation will occur relative to the XY axis. When the mouse is moved toward the outer perimeter of the window, the XY will change to a Z. Holding the button down while the Z is visible will rotate the drawing on the Z axis.

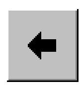
Converts the left mouse button to a pan function. Hold down on the button while moving the mouse to pan. Holding down the mouse wheel will also serve as a pan function in any of the above modes.



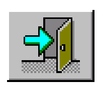
Toggles shading on and off.



Restores the graphics to plan view.



Reverses the effects of all operations performed on the TIN and reverts it back to its original status.



This icon exits the routine. If the TIN has been modified, you will be prompted to save.

Pulldown Menu Location: Surface
Keyboard Command: TINUTIL
Prerequisite: 3D Faces, a TIN file or a DXF file.
File Name: \lsp\tri4.arx

Two Triangulation Surface Volumes

Volumes By Triangulation is an alternative volume method that compares two triangulation networks. This method is different from the grid based volume routines (*Volumes By Layer, One Surface Volumes, Two Surface Volumes, Stockpile Volumes, etc.*) and the cross section volume routine (*Calculate Section Volume*). Volumes by Triangulation calculates faster in most cases than the other methods, and it is the most accurate because it uses true TIN to TIN prismoidal volumes. This added accuracy in general is very small. The grid resolution is usually sufficient to model the surface for the grid based volumes. The Volume By Triangulation accuracy applies well when there is a feature like a 5 foot wide ditch. Then the grid resolution would need to be less than 5 feet to model the ditch which might be difficult on a large site.

The disadvantage to this routine is that it lacks the output options that help the analysis of the volume such as Difference Contours. Also Volumes by Triangulation does no extrapolation and stops calculating volume at the

perimeter of the smaller of the two triangulation networks. Volumes By Triangulation is better when used with point data instead of contour data because contour data requires triangulating all the contour polylines as breaklines which creates a large triangulation network and is slower.

The triangulation networks to compare are defined in .tin or .flt files that are created by *Triangulate & Contour* with the Write Triangulation File option. Note that while both file formats are supported, the newer binary triangulation file format (.tin) is twice as fast to load and save, and half the size, of the .flt triangulation file format. For this reason, the .tin file format is recommended. Before using this command, run *Triangulate & Contour* twice to create an triangulation (.TIN or .FLT) file for each surface. The volume calculation is limited by either the extent of the triangulation networks or by an inclusion/exclusion perimeter(s). These perimeters must be closed polylines.

Output data includes area, tons by density, average thickness, shrink and swell, ratio, and total volume.

Prompts

Select EXISTING surface Tmesh File
Choose an .flt or .tin file
Select final elevation or surface TIN [Elevation/<TIN>]? *press Enter*
Select FINAL surface Tmesh File Dialog
Choose another .flt or .tin file.
Choose Predefined Boundary Dialog Choose none.
Select Inclusion polylines *select objects*
Select Exclusion polylines *select objects*
Volume Report Choose viewing option
Comparing Triangulation files: C:\SCAD2006\DATA\TRI1.FLT
and C:\SCAD2006\DATA\TRI2.FLT
Cut volume: 66891.35 C.F., 2477.46 C.Y.
Fill volume: 43458.01 C.F., 1609.56 C.Y.

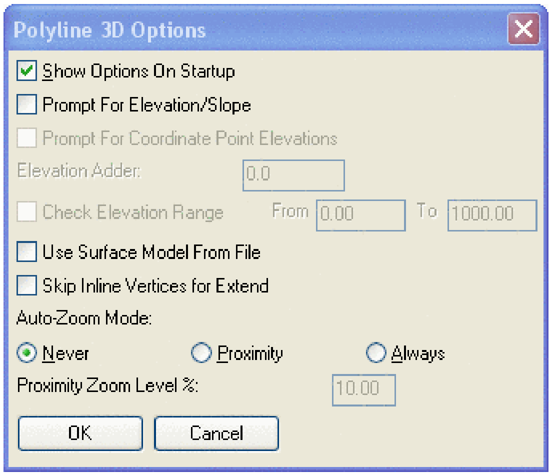
Keyboard Command: trivol
Prerequisite: Two .flt or .tin files

Draw 3DPoly Perimeter

This command draws a 3D polyline on the PERIMETER layer. This is one way to generate the polyline that is required by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines. In these routines, this polyline is used as the inclusion perimeter for volumes. If you are using Carlson points to define the polyline, make sure they are present in the drawing at their real Z elevation, and then set your AutoCAD Object Snap to Insert prior to running this routine. Alternately, you can use point numbers from the current coordinate (.CRD) file. A third option is to obtain the elevations of picked points from a specified surface model. You have a further option to be prompted for each elevation, thus overriding the values found from the included points.

Prompts

If set to display, the 3D Poly options dialog will appear, and then you will be prompted for points to use to draw the 3DPoly Perimeter. Standard Carlson point number input or screen picks using AutoCAD selection methods are valid.



Dialog Options:

Prompt for elevations (.XY filter) (Yes/<No>)?: *Y* Using the .XY filter allows the user to pick the X and Y coordinate from the screen and type in the elevation. If you use the No response then the Z coordinate of the point picked will be applied.
Use surface model from file [<Yes>/No]: *Y* With this option, a surface file is specified, and then with each screen pick, the surface elevation is determined. If Prompt for elevations is set to No, the surface elevation is applied to the polyline vertex. If set to Yes, the surface elevation is displayed as the default, and can be accepted by pressing Enter, or a different elevation can be typed in instead.

Pick point or point number: *pick a point or type a point number*
Arc/Close/Undo/Pick point or point number: *15* This is a point number from the current coordinate (.CRD) file.

Note that if the response to Use Surface model from file is *Yes*, the elevation used is not the point elevation from the coordinate file (.CRD), but the elevation interpolated from the surface.

Arc/Close/Undo/Pick point or point number: *press Enter*
Draw another 3D polyline [Yes/<No>]? *press Enter* Pressing Enter ends the command.

Pulldown Menu Location: Surface >> Stockpile/Pond/Pit Volumes
Keyboard Command: 3dperim
Prerequisite: None
File Name: \lsp\3dpline.lsp

Draw 3DPoly Base Breakline

This command draws a 3D polyline in the BASE_BREAKLINE layer. This polyline is used by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines to model the base surface. You may want to set your AutoCAD Object Snap prior to running this routine so that you obtain the elevations of existing points while creating the 3D polyline. Besides picking and entering the points, you can also use point numbers from the current

coordinate (.CRD) file.

This routine functions identically to the *Draw 3DPoly Perimeter* command, only placing the resulting 3D polyline on a different layer.

Pulldown Menu Location: Surface >> Stockpile/Pond/Pit Volumes
Keyboard Command: 3dbase
Prerequisite: None
File Name: \lsp\3dpline.lsp

Calculate Stockpile Volume

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is above the perimeter elevation, such as in the case of a stockpile of material. The complimentary command, *Calculate Pond/Pit Volume,* is for the opposite situation, in which the entire volume to be calculated is below the elevation of the perimeter.

The program internally computes BASE and FINAL grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select another 3D polyline to use. The 3D polyline perimeter can be drawn with the *Draw 3D Polyline Perimeter* command before using this routine.

The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the BASE surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The FINAL surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, **but not including the BASE_BREAKLINE polylines**.These features are used only in computing the BASE surface.

You have the option of setting the resolution of the grids.

The *Make 3D Grid File* and *Two Grid Surface Volumes* commands, used in combination, are an alternative to this command, and in any situation in which there are cut and fill volumes between the surfaces, that combination must be used to generate accurate results.

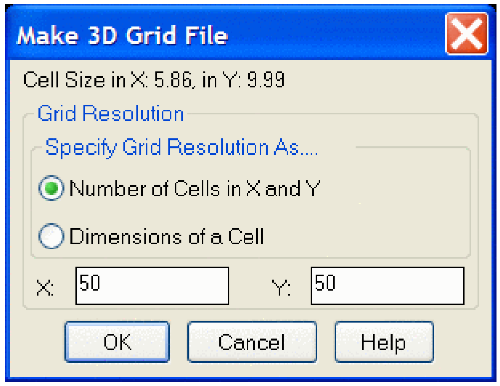
Prompts

Material density lbs/ft^3 (Enter for none): *enter a material density in lbs per cubic foot, or press Enter for none*

Ignore Zero Elevations [<Yes>/No]?

Select stockpile entities and perimeter.
Select objects: *pick the objects that define the stockpile and the 3D polyline perimeter*

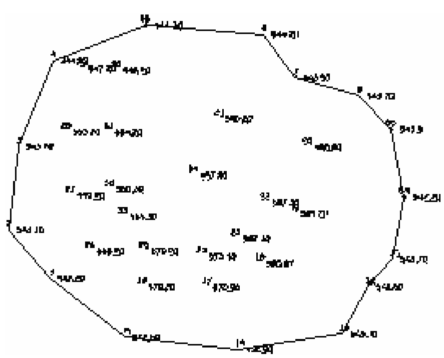
Select stockpile perimeter polyline:



Make Grid File dialog Set the resolution and then click OK.

<u>Sample volume report</u>

```
 Volume report
Lower left grid corner: 15965.45,12657.05
Upper right grid corner: 16269.40,12906.29
X grid resolution: 50, Y grid resolution: 50
X grid cell size: 6.08, Y grid cell size: 4.98
Stockpile volume: 1191674.87825 cubic ft, 44136.107 cubic yards
```



Stockpile defined by points and a 3D polyline perimeter
Window these objects to obtain the volume report

Pulldown Menu Location: Surface >> Stockpile/Pond/Pit Volumes
Keyboard Command: stockvol
Prerequisite: Data representing the stockpile surface and a 3D polyline representing the perimeter of the stockpile.
File Names: \lsp\stockvol.lsp, \lsp\volcalc.arx

Calculate Pond/Pit Volume

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is below the perimeter elevation, such as in the case of a pond or pit. The complimentary command, *Calculate Stockpile Volume*, is for the opposite situation, in which the entire volume to be calculated is above the elevation of the perimeter.

The program internally computes BASE and FINAL grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select another 3D polyline to use. The 3D polyline perimeter can be drawn with the *Draw 3D Polyline Perimeter* command before using this routine.

The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the BASE surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The FINAL surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, **but not including the BASE_BREAKLINE polylines**.These features are used only in computing the BASE surface.

You have the option of setting the resolution of the grids.

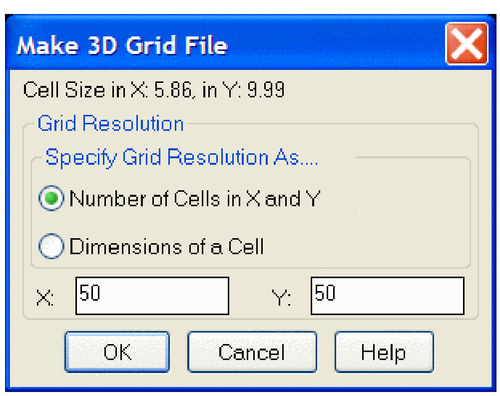
The *Make 3D Grid File* and *Two Grid Surface Volumes* commands, used in combination, are an alternative to this command, and in any situation in which there are cut and fill volumes between the surfaces, that combination must be used to generate accurate results.

Prompts

Ignore Zero Elevations [<Yes>/No]?

Select Pond/Pit entities and perimeter.
Select objects: *pick the objects that define the surface and the 3D polyline perimeter*

Select Pond/Pit perimeter polyline:



Make Grid File dialog Set the resolution and then click OK.

Pulldown Menu Location: Surface >> Stockpile/Pond/Pit Volumes
Keyboard Command: pitvol
Prerequisite: Data representing the pond/pit surface and a 3D polyline representing the perimeter of the pond/pit.
File Names: \lsp\stockvol.lsp, \lsp\volcalc.arx

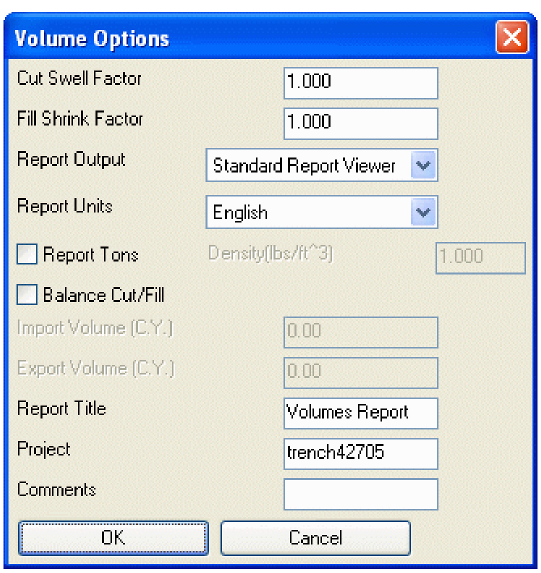
Calculate Total Volumes

Use this command to report total volume calculations within the site boundary polyline. The report includes the cut and fill quantities as well as the strata and topsoil quantities if the site has strata and topsoil defined. Besides reporting the total quantities for the site boundary, Area Of Interest polylines can be used to report quantities within named perimeters.

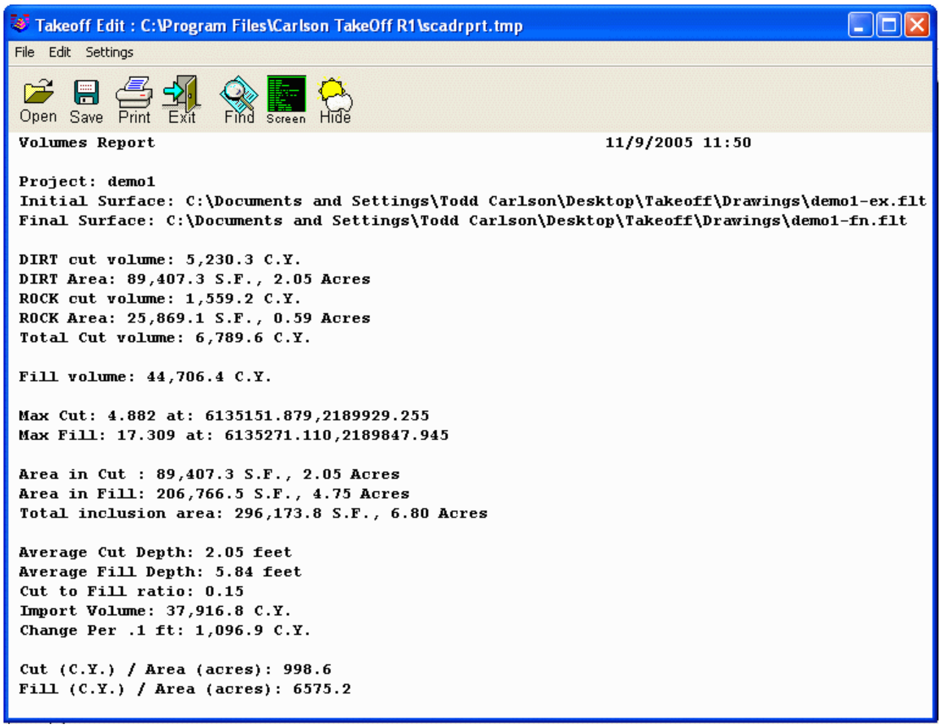
Before running this command, the existing and design surfaces must be created and the boundary polyline must be assigned. Also, the strata surfaces, topsoil and Area Of Interest polylines need to be set before this command if those features are to be reported.

The Volume Options dialog box shown here offers options for the final report. Here you can select four different types of reports: Standard Report Viewer, Custom Report Formatter, Expanded Auto Format, and Compressed Auto Format. The Cut Swell Factor is multiplied by the cut volume and the Fill Shrink Factor is multiplied by the fill volume. The Report Units setting chooses between English and Metric quantities for the report. In Drawing Setup in Takeoff, you set the drawing units as either English or Metric. The Report Units will default to match the drawing units but you can change the Report Units to the other mode and the program will apply the conversion between English and Metric for the report. So you can have a drawing in English units and create a report with Metric quantities.

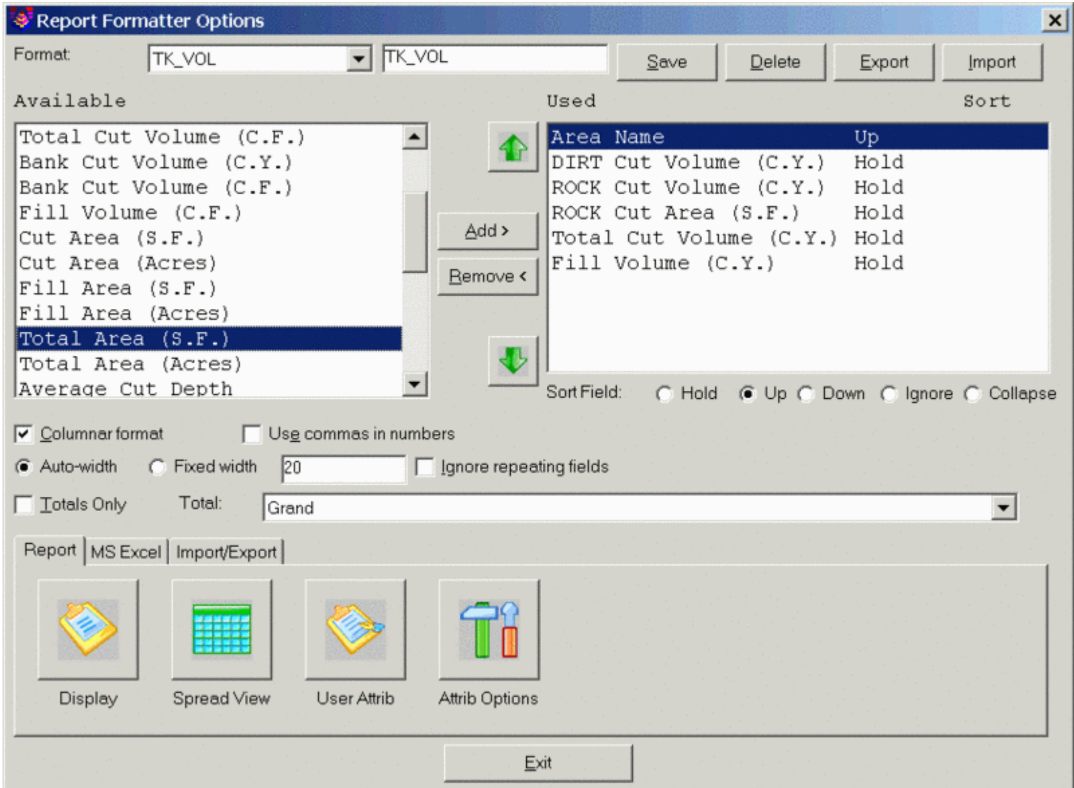
Note: As the quantities are calculated within each area, the area is hatched with a solid fill as a visual verification that the right area is being processed.



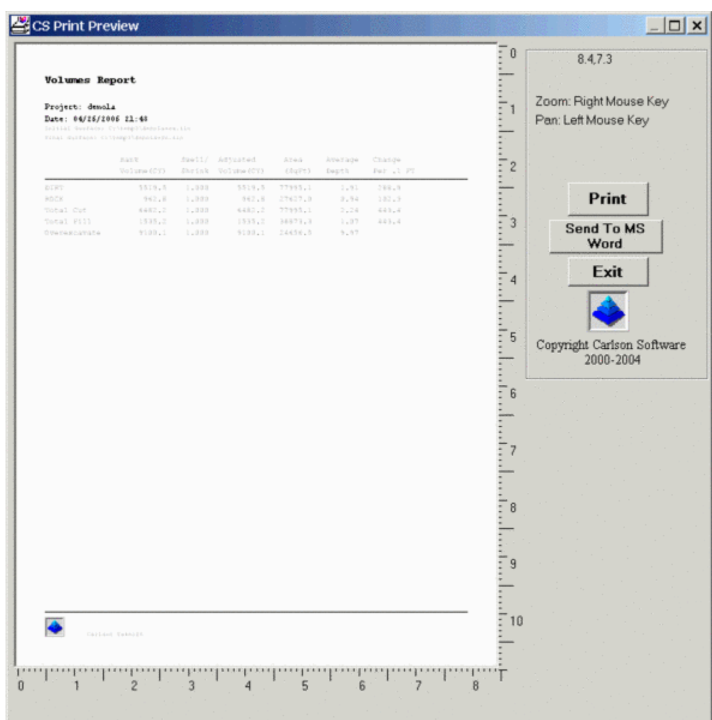
Shown here is an example of a Standard Report Viewer.



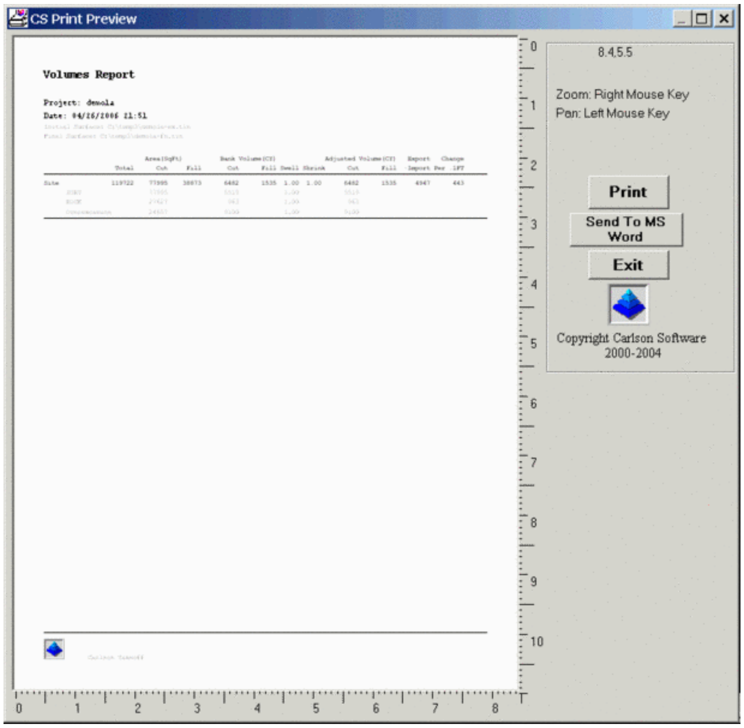
Use Customs Report Formatter to customize or "user define" the reporting options. The Report Formatter Options dialog box shown here offers a variety of output options including Excel. You can choose the fields to report from the Available list and set their report order under the Used list.



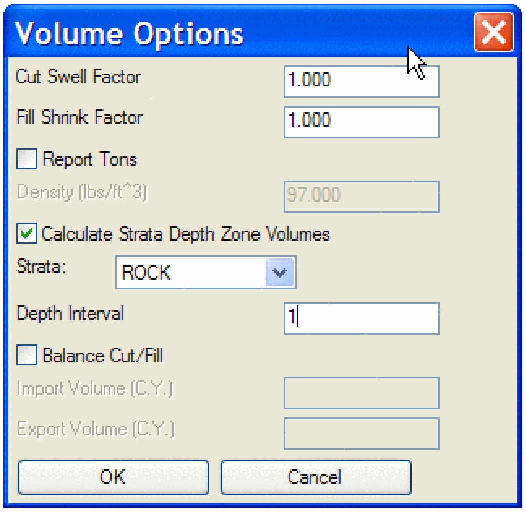
The Expanded Auto Format is shown in this report preview. You can also send the report to MS Word in this command.



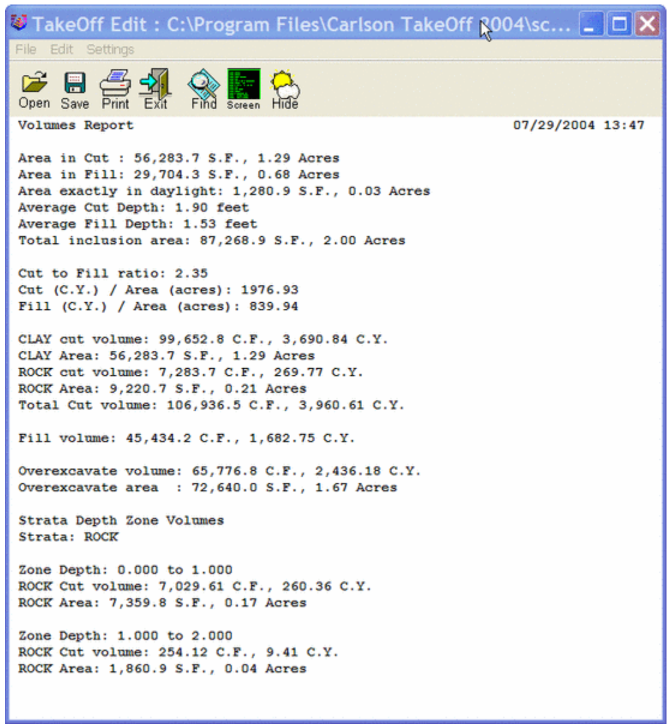
The Compressed Auto Format puts all the area information on one row.



If drillholes have been located on the drawing and strata types and depths have been defined, a calculate Strata Depth Zones Volume option
becomes available as shown here.



Shown here is an example of the report if strata depth intervals have been defined.



Volumes Report 07/29/2004 13:47

Area in Cut : 56,283.7 S.F., 1.29 Acres
Area in Fill: 29,704.3 S.F., 0.68 Acres
Area exactly in daylight: 1,280.9 S.F., 0.03 Acres
Average Cut Depth: 1.90 feet
Average Fill Depth: 1.53 feet
Total inclusion area: 87,268.9 S.F., 2.00 Acres

Cut to Fill ratio: 2.35
Cut (C.Y.) / Area (acres): 1976.93
Fill (C.Y.) / Area (acres): 839.94

CLAY cut volume: 99,652.8 C.F., 3,690.84 C.Y.
CLAY Area: 56,283.7 S.F., 1.29 Acres
ROCK cut volume: 7,283.7 C.F., 269.77 C.Y.
ROCK Area: 9,220.7 S.F., 0.21 Acres
Total Cut volume: 106,936.5 C.F., 3,960.61 C.Y.

Fill volume: 45,434.2 C.F., 1,682.75 C.Y.

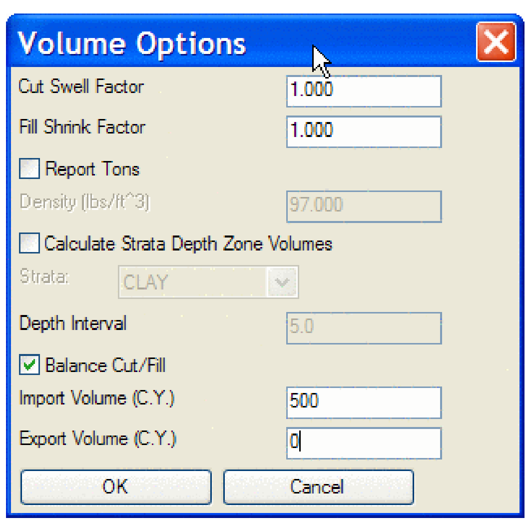
Overexcavate volume: 65,776.8 C.F., 2,436.18 C.Y.
Overexcavate area : 72,640.0 S.F., 1.67 Acres

Strata Depth Zone Volumes
Strata: ROCK

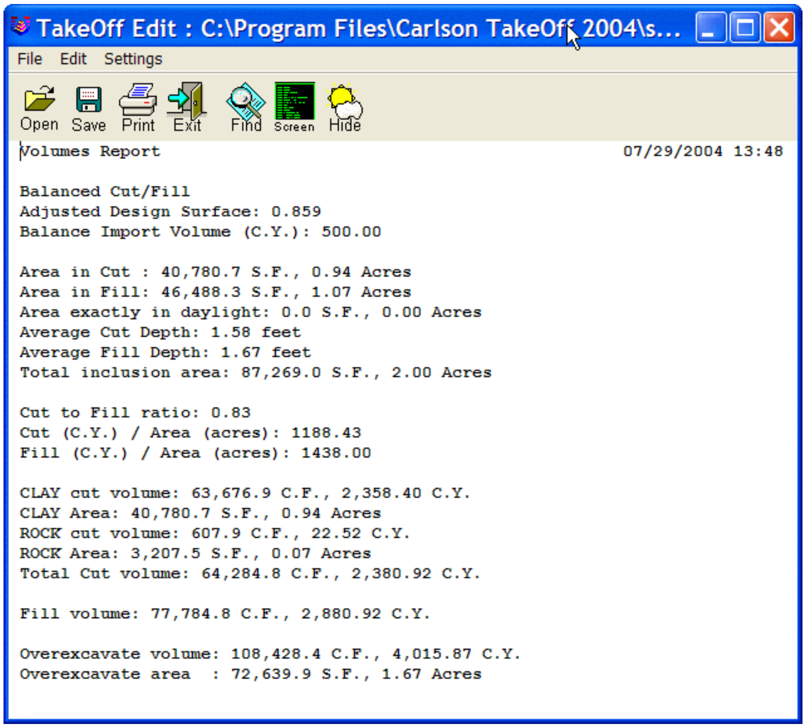
Zone Depth: 0.000 to 1.000
ROCK Cut volume: 7,029.61 C.F., 260.36 C.Y.
ROCK Area: 7,359.8 S.F., 0.17 Acres

Zone Depth: 1.000 to 2.000
ROCK Cut volume: 254.12 C.F., 9.41 C.Y.
ROCK Area: 1,860.9 S.F., 0.04 Acres

The Balance Cut/Fill option shown here allows an import or export volume in cubic yards option. Use these options if waste material is available or needed elsewhere. If this option is used the resulting report indicates the vertical movement of the site needed to satisfy the balance option.



Shown here is a report with a 500 CY importation of material and suggests that the site be vertically raised 0.859 feet.



If the adjusted surface is satisfactory, Carlson Takeoff offers the option to save the adjusted surface as shown here in the Balance Cut/Fill dialog box.

Prerequisite: Existing and design surfaces and a boundary polyline
Keyboard Command: tin_volume

Calculate Volumes Inside Perimeter

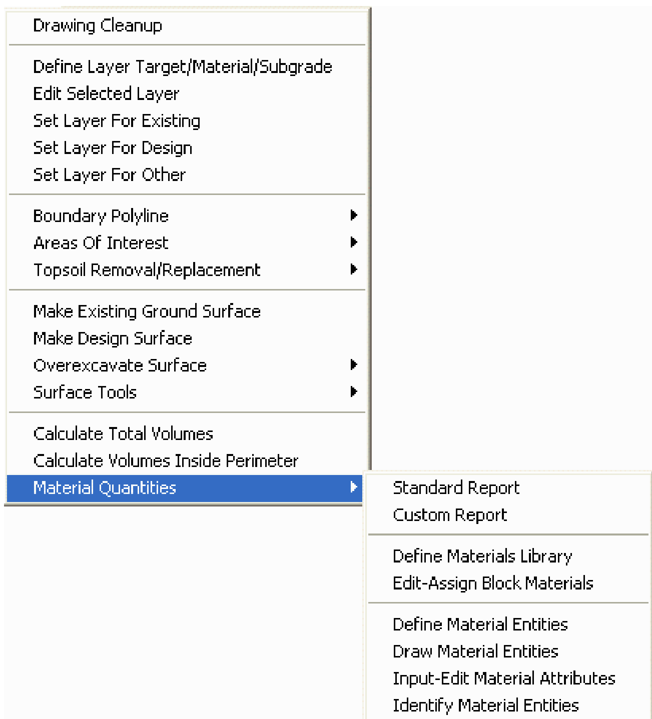
Use this command to create volume reports inside the selected closed perimeter polyline. The same reporting options are available for this command as are for the Calculate Total Volumes command.

Keyboard Command: tin_volume2
Prerequisite: Existing and Design surfaces and a closed perimeter polyline

Material Quantities

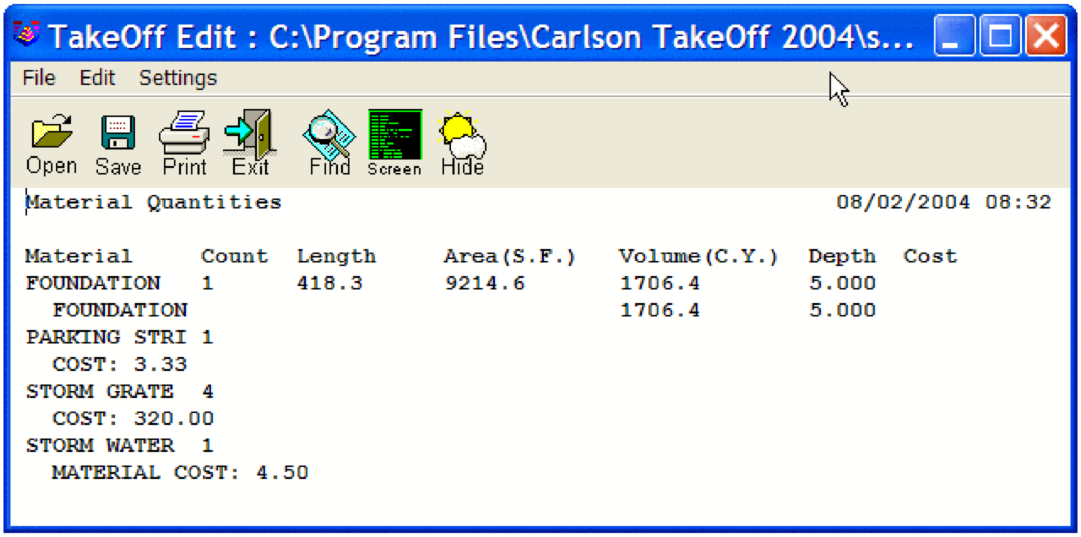
The Material Quantities flyout shown here offers many options for quantity reporting including the option for user defined attributes. Entities with attributes can be drawn, edited, and identified. Standard and custom report options are also available.

Material Quantities are counted from the entities in the drawing. Several entity properties can be reported including entity count, length, area and volume. Also user-defined attributes can be assigned to the entities and reported. The type of material for each entity is determined by the layer for the entity. In the Define Layer Target/Material/Subgrade, you can assign the material types by layer.



Standard Report

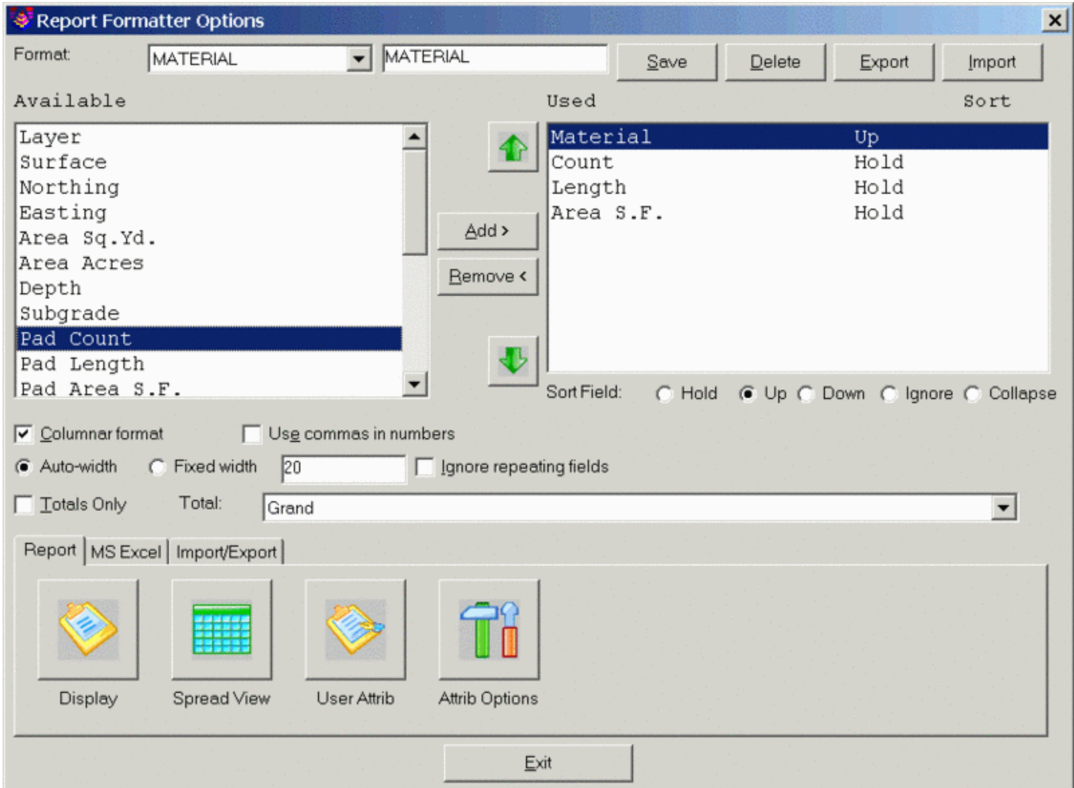
Use this command to display all or a selected set of material quantities and user-defined information with the standard Carlson Takeoff report format shown here.



Prerequisite: Defined materials
Keyboard Command: materials_report2

Custom Report

Use this command to customize or "user define" the reporting options. This command first prompts whether to report quantities for all the entities in the drawing or selected entities. Then if the drawing contains Areas Of Interest polylines, there is an option report quantities by these areas which adds the area name to the available report fields to allow sorting and grouping by area name. The Report Formatter Options dialog box shown here offers a variety of output options. You can choose the fields to report from the Available list and put them in report order under the Used list.

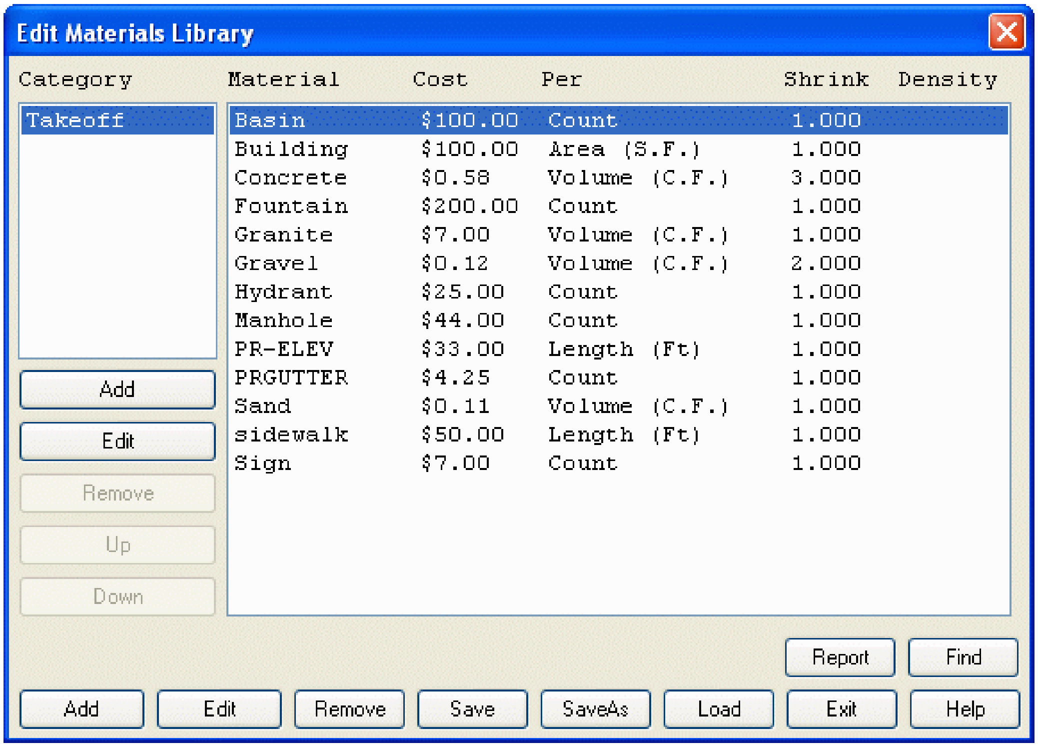


Selecting the Display option shows the report in the standard Carlson Takeoff report viewer. Reports can be exported to an Excel spreadsheet as well.

Prerequisite: Defined materials
Keyboard Command: materials_report

Define Materials Library

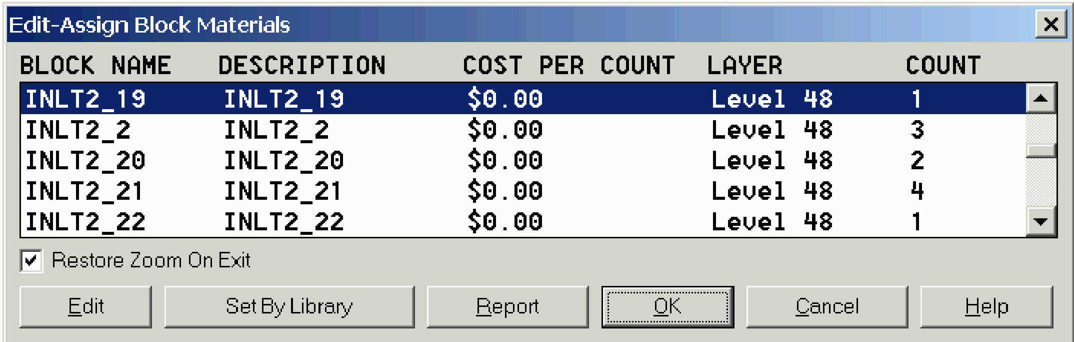
Define Materials Library allows you to Add, Remove, Load, Save, and Report a list of material costs. Costs can be set per area, count, volume, ton, or length by using the Edit function at the bottom of the dialog. The left side of the dialog can be used to set categories for different material costs.



Prerequisite: pricing for materials
Keyboard Command: define_tk_materials

Edit-Assign Block Materials

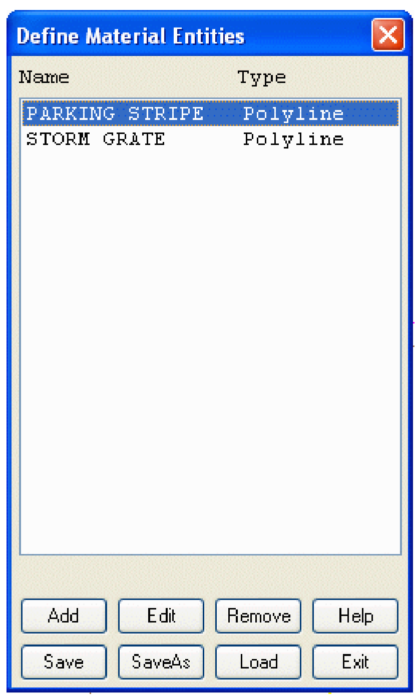
This command scans the current drawing to find and report block/symbol names and their count. For example, when the drawing contains different symbols for different types of utilities, this command identifies each type of symbol and the number. From this command, you can set the Description and Cost of the block by using the Edit button. You can also set the Description and Cost by predefined Materials by using the Set By Library button. When a block name is highlighted from the list, the drawing is zoomed to the location of one of those blocks so that you can see what it looks like. To Report these materials as part of the Standard Report, check on Include Materials Quantities Report in the Edit Materials dialog of the block layer found in the Define Layer Target/Material/Subgrade command. You can also just click on the Report button for a simple report.



Prerequisite: Blocks
Keyboard Command: edit_all_blocks

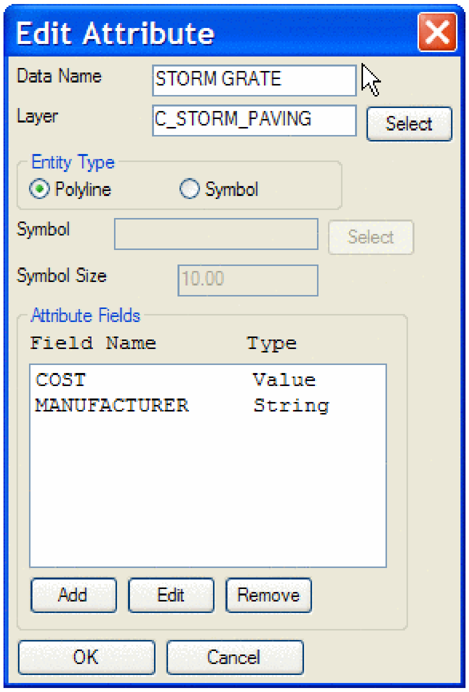
Define Material Attributes

Use this command to define all the material attributes that will be assigned to objects in the drawing for reporting purposes. The Define Attribute dialog box shown here allows the user to "Add", "Edit", or "Remove" attributes and save the definitions for later use. Simply "Load" a saved attribute definition file with the "tkd" extension for future use.

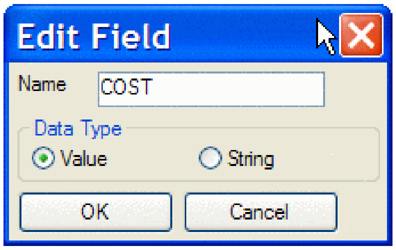


Selecting the Add or Edit options produce the edit attribute dialog box shown here. Use this command to define the Data name and the layer the objects currently reside on and the layer that future objects will be drawn on.

Two entity types can be used, polyline data or point data. If the symbol option is selected the user has the option of which symbol will represent the object. Attribute fields must be defined for material reporting.



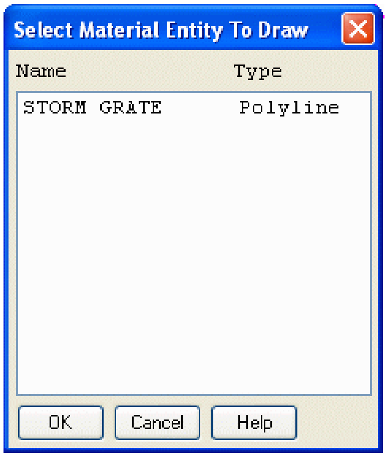
Selecting the Add or Edit button on the Edit Attribute dialog box brings up the Edit Field dialog box shown here. Use this dialog to define the field name and type. If the Value option is selected, only numeric values will be allowed when prompted. If the String option is selected, the user will have the ability to type in a text message when prompted.



Prerequisite: attributes
Keyboard Command: define_tk_data

Draw Materials Entities

Use this command to apply attribute data to objects as you draw or digitize them. Select the predefined attribute type to draw from the list available in the Select Attribute to Draw dialog box shown here.

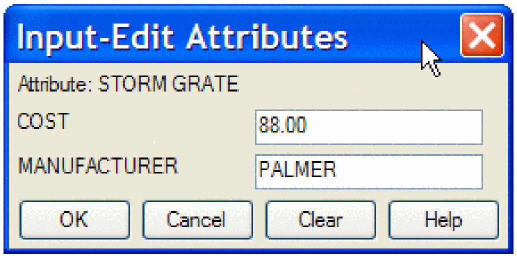


The command line will prompt the user to pick the points of the desired location of the object and allow the attribute data fields to be filled out upon completion or each "enter".

Prerequisite: defined attributes
Keyboard Command: draw_tk_data

Input-Edit Material Attributes

Use this command to assign predefined attribute information to an object already existing in the drawing. The command line prompt will require the user to select the object that attribute information is to be applied, and offer the Input-Edit Attribute dialog box shown here. This dialog box will display all predefined fields for that particular attribute type.



Prerequisite: predefined attribute information
Keyboard Command: edit_tk_data

Identify Materials Entities

Use this command to display all the objects that have attribute data assignments. The user will have the options of selecting the objects by picking them individually or by searching the entire drawing database. The objects that have attribute information assignments will "highlight" on the screen and the command line will display the attribute information.

Prerequisite: attributes
Keyboard Command: id_tk_data

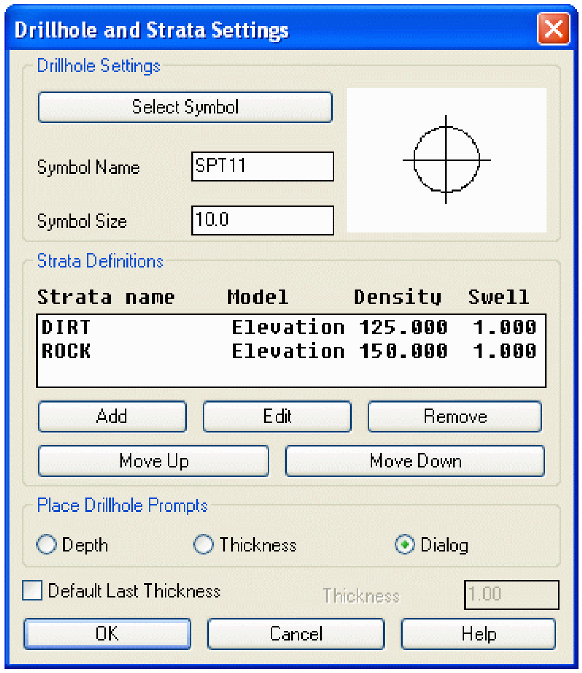
Drillholes Menu 9

Drillhole Strata Settings

This command selects drillhole symbols, defines strata, and determines how you place drillholes.

Note: The order in which the Strata are defined in the Strata Definitions list will be the default order for the strata when you create new drillholes through **Place Drillhole**.

The dialog box below shows the layout of the Drillhole and Strata Settings.



- **Select Symbol**: Select a symbol to represent the drillhole location on the screen.

- **Symbol Name:** This name corresponds to the symbol selected.

- **Symbol Size:** This field can be edited to adjust the symbols size displayed on the screen.

- **Strata Definitions:** This is not directly editable. Select the Strata you are interested by highlighting it, then select the Edit button.

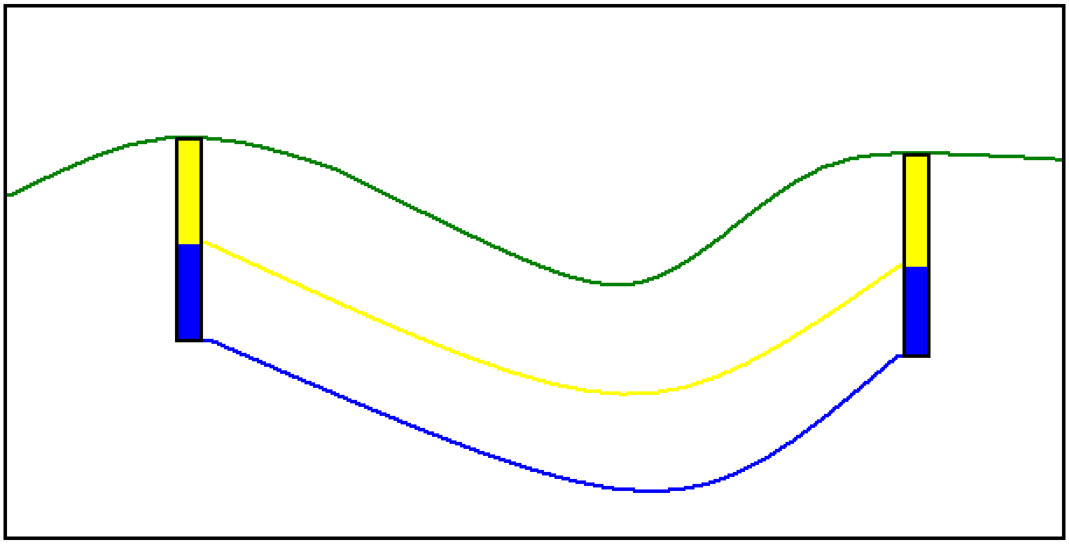
- **Add:** This option adds additional strata to the available Strata name list. See Edit Strata dialog box below.

- **Edit:** Similar to Add, this option is available to make changes to the Strata, including adding a swell factor.

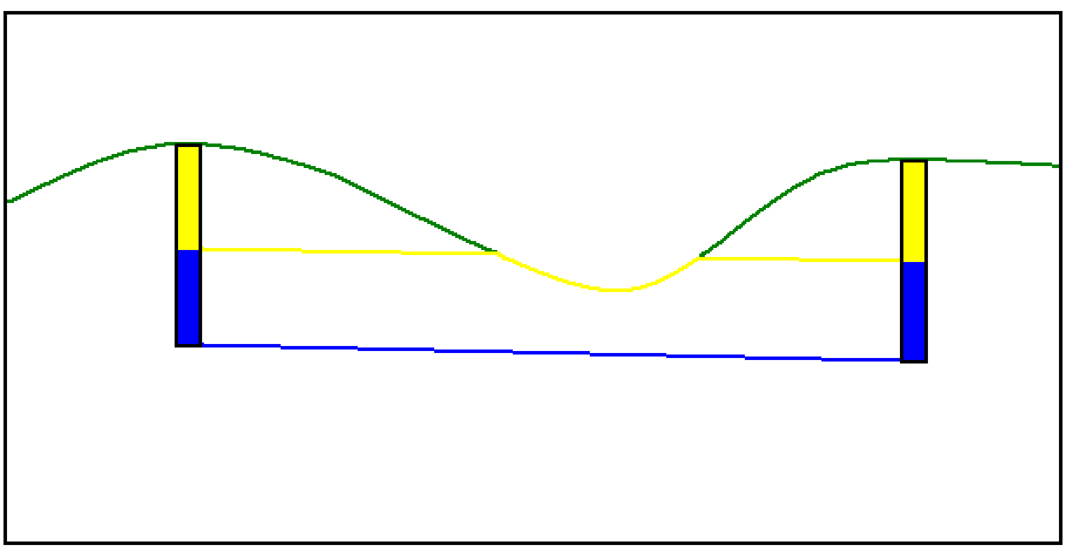
• **Strata Name:** The name of the strata.

Density (lbs/ft^3): The Strata Density field is the default density used to calculate strata tons. Density is strata-specific.

• **By Depth:** This option will generate a strata surface by modeling the strata depth values in the drill-holes. This strata surface will follow the existing ground surface at the model depth.



• **By Strata Elevation:** This option will generate a strata model that connects strata irrespective of the upper surface elevation changes.



- **Remove:** This will remove a strata name from the available strata.

- **Move Up:** This option will move the selected strata name up one place in the strata name list.

- **Move Down:** This option will move the selected strata name down one place in the strata name list.

- **By Strata Elev:** This method will generate a strata surface by modeling the strata elevation values from the drillholes. This strata surface is independent of the existing ground surface.

- **Place Drillhole Prompts:** If Depth is selected, then when you run Place Drillholes you will be prompted for the depth of each strata in your drillhole. If Thickness is selected, you will be prompted for the thickness of each strata. If Dialog is selected, you will go straight into the Place Drillhole dialog when you create a drillhole.

- **Default Last Thickness:** Will set the thickness of your bottom strata to the same amount for all your drillholes.

Keyboard Command: tk_chdef
Prerequisite: strata information

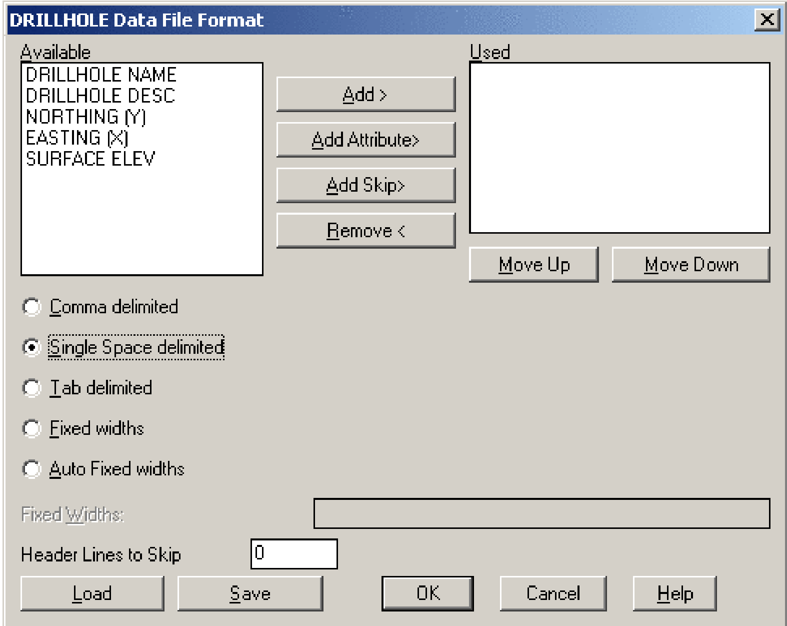
Drillhole Import

Function

This command allows you import existing drillhole files. When you select Drillhole Import from the Drillhole menu, a command prompt shows:

"Use separate drillhole and strata files [Yes/<No>]?" If you have two separate files, one with strata info, and the other file has drillhole locations, select Yes. If you enter Yes, the dialog box below appears.

This command creates drillholes from the data contained in text files. Currently there are many company-specific formats. A Drillhole Data File Formatter that is flexible to handle almost any drillhole text file format is below. The format to use is chosen in the dialog shown here.



The import text can have comma delimited, space delimited or fixed width columns of data. All the data for a record should be on one row. For the fixed width format, choose the Fixed Width toggle and then enter the column numbers separated by spaces in the edit box. For example, "8 15 24 32".

The Custom format can import all the drillhole and strata data from one text file or the drillhole data from one file and the strata data from another file. The method to use is set at the Use separate drillhole and strata files prompt.

Use the following commands to prepare a file format that will match the ∗.imp imported file.

- **Add:** Moves the selected entry from Available to Used.
- **Add Attribute:** Allows user input attributes into the Used section.
- **Add Skip:** Adds a "Skip" place holder in the Used List
- **Remove:** Moves a selected item from Used to Available list.
- **Move Up:** Moves the selected item up one place in the list.
- **Move Down:** Moves the selected item down one place in the list.
- **Comma Delimited:** Select this if your ∗.imp file has commas separating each field.
- **Single space delimited:** Select this if your ∗.imp has a space separating each field.
- **Tab delimited:** Select this if your ∗,imp file has tabs separating each field.
- **Fixed widths:** Select this if your ∗.imp has a defined width of space separating each field.

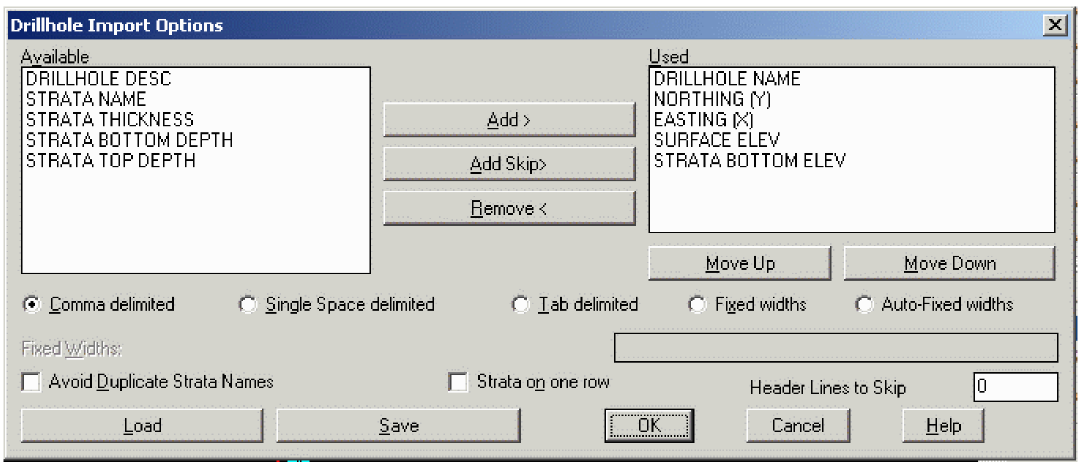
- **Auto Fixed widths:** Select this to automatically determine the fixed widths that separate each field in the ∗.imp file.

- **Header Lines to Skip:** If your ∗.imp file has header lines, enter the number of header lines here.

- **Load:** Takes you to select/brows for your ∗.imp file.

- **Save:** This command will save your imported file as a ∗.imp file.

The dialog box below details the drillhole import options.



In addition to the previously listed import commands above, this dialog box also has the following prompts:

- **Avoid Duplicate Strata Names:** Select this to prevent have more than one strata with the same name.

- **Strata on one row:** Select this option if all of your strata info is on one row.

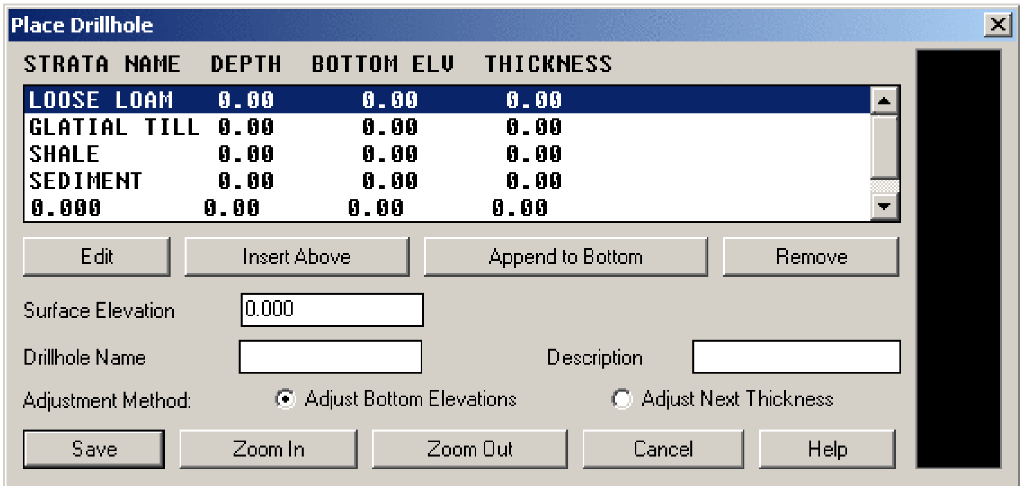
Keyboard Command: tk_chimport
Prerequisite: drillhole files

Place Drillhole

This command allows you to screen pick or enter coordinates the placement of a drillhole.

Go to **Drillhole/Strata Settings**, **Place Drillhole Prompts**, to determine how you would like to be prompted. When you select **Place Drillhole** from the **Drillhole menu**, the command line prompt shows:

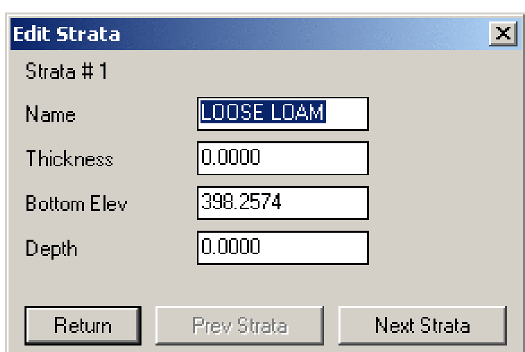
"Pick Drillhole Location:" Type in x-y coordinates or move your pointer around the screen to pick the placement of the drillhole. If you are in Dialog Mode defined in **Drillhole/Strata Settings**, once a location was picked on the screen, the following dialog box appears:



Place DrillHoles generates drillholes in the drawing that are required to run strata surface application routines. Each drillhole consists of a surface elevation, strata, and optional description(s). Every strata has a name, bottom elevation, thickness. Within a drillhole, the strata names must be unique, but each real-world strata should have the same strata name across all the drillholes. This is because strata surface applications connects together the strata with the same name.

The drillhole data can be entered in the dialog shown here, or if Depth or Thickness Mode is selected under **Drillhole/Strata Settings**, then the data can be entered in on the command line when you place each drillhole. Make sure to specify the surface elevation and drillhole description. While in Dialog Mode or to change data, use the Edit and Insert/append buttons to enter strata data. The symbols are defined in DrillHole/Strata Settings and drillhole may be changed in Edit DrillHole. Pick Save when done and a drillhole symbol is drawn.

- **Edit:** Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.



When placing drillholes, every strata must be assigned a bottom elevation and a thickness. The bottom elevation is

the elevation of the bottom of the strata. There are different methods for entering this information.

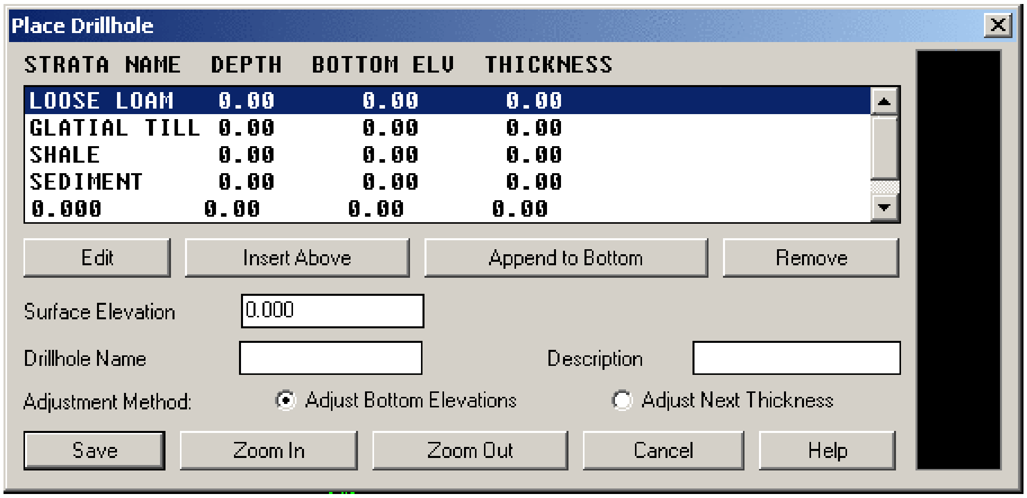
- **Insert Above:** To add a Strata above the highlighted strata name.
- **Append to Bottom:** To add a strata to the bottom of the available strata name list.
- **Remove:** Removes a strata from the available Strata Name list.
- **Surface Elevation:** This field can be set by you to establish th surface elevation of the drillhole.
- **Drillhole Name:** The name of the drillhole
- **Description:** Drillhole descriptions are intended for storing of drillhole specific information in the drillhole. One general drillhole description is predefined and user may define any number of specific drillhole descriptions. Typical additional descriptions are DRILLER, DATE, TOWNSHIP, and etc. You will be prompted for values of these descriptions in Place DrillHole.
- **Adjust Bottom Elevations:** Will make adjustments to the bottom elevation based on thickness changes.
- **Adjust Next Thickness:** Will adjust the next thickness to hold the bottom elevation unchanged.
- **Save:** This command saves this drillhole as listed.
- **Zoom In:** This increases the magnification of the black view window, cross-section view of the drillhole.
- **Zoom Out:** This decreases the magnification of the black view window, cross-section view of the drillhole.
- **Cancel:** Ends Drillhole placement routine without making changes.

Keyboard Command: tk_chplace
Prerequisite: drillhole information

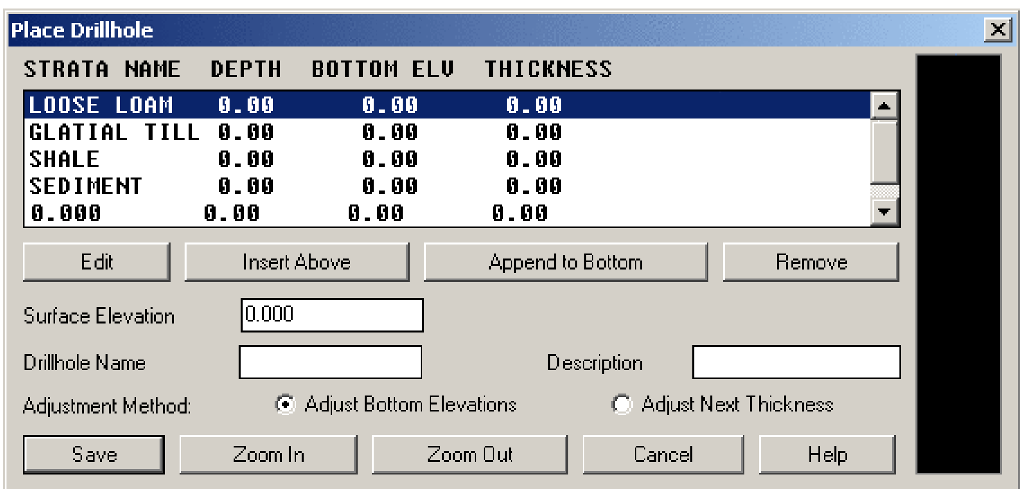
Edit Drillhole

This command allows you to screen pick an existing drillhole and edit its properties. When you select **Edit Drillhole** from the **Drillhole menu**, a command prompt shows:

"Select Drillhole to edit:" Move your pointer around the screen to pick the drillhole you want to edit. Once a drillhole is picked on the screen, the following dialog box appears:



- **Edit:** Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.



- **Insert Above:** To add a Strata above the highlighted strata name.
- **Append to Bottom:** To add a strata to the bottom of the available strata name list.
- **Remove:** Removes a strata from the available Strata Name list.
- **Surface Elevation:** This field can be set by you to establish th surface elevation of the drillhole.
- **Drillhole Name:** The name of the drillhole
- **Description:** The screen display description of the drillhole

- **Adjust Bottom Elevations:** Will make adjustments to the bottom elevation based on thickness changes.

- **Adjust Next Thickness:** Will adjust the next thickness to hold the bottom elevation unchanged.

- **Save:** This command saves this drillhole as listed.

- **Zoom In:** This increases the magnification of the black view window, cross-section view of the drillhole.

- **Zoom Out:** This decreases the magnification of the black view window, cross-section view of the drillhole.

- **Cancel:** Ends Drillhole placement routine without making changes.

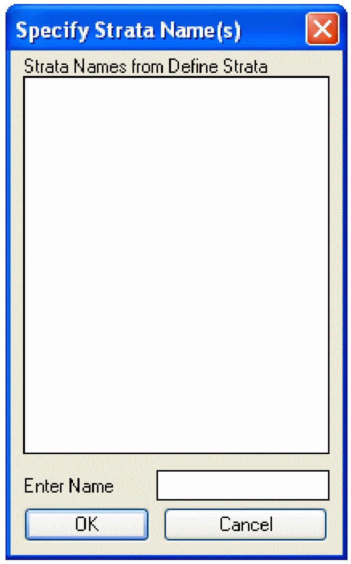
Keyboard Command: tk_chedit
Prerequisite: drillhole information

Strata Polylines

Strata Polylines define strata elevation or thickness along linework instead of a single point like Place Drillhole. Linework defined as Strata Polylines are incorporated with Drillhole Data to create surface models. Note: Strata surface models can not be made exclusively from Strata polylines, some drillholes need to be placed as well.

Tag Strata Polylines

This command allows the user to select polylines that define a Strata. Pick the Strata from the list or type in the name in the Enter Name field. Any Strata you enter in must match a strata defined in Drillhole/Strata Settings in order for the surface to be created.



After selecting a Strata and pressing enter you will be prompted for the type of polyline.

Type of strata polyline [<Elevation>/Thickness]?

Elevation signifies that the Z value for the polyline(s) you are about to select represent the bottom elevation of the previously selected strata. Thickness means that the Z value represents thickness of the strata. Choose one of these options and select the polylines.

Prerequisite: Drillhole/Strata Settings, desired polylines
Keyboard Command: stratatag

Highlight Strata Polylines

This command allows users to identify Strata Polylines by either picking on a polyline(s) or by searching the entire drawing. The command will then dash the polyline in the plan view.

Prerequisite: Tag Strata Polylines
Keyboard Command: highlight_strata_pl

Identify Strata Polylines

This command allows users to identify topsoil polylines by picking on a polyline. The command will then report the Strata name and Type (either Elevation or Thickness).

Prerequisite: Tag Strata Polylines
Keyboard Command: strataid

Untag Strata Polylines

This command allows the user to remove previously tagged Strata Polylines so that they are not included in the strata model.

Prerequisite: Tag Strata Polylines
Keyboard Command: strатauntag

Drillhole Reports

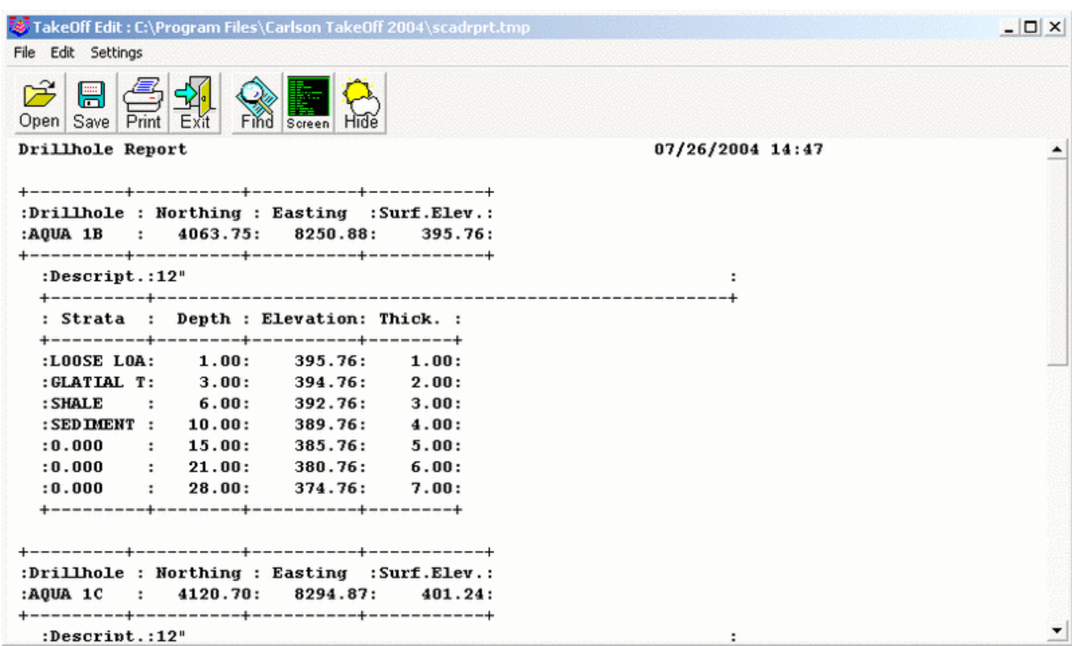
This command allows you to generate a report of selected drillholes. When you select **Reports** from the **Drillhole** menu, a sub-menu choice of **Standard Drillhole Report** or **Custom Drillhole Report**, is displayed.

Standard Drillhole Report

If this is selected, several prompts are asked at the command line. They are as follows:

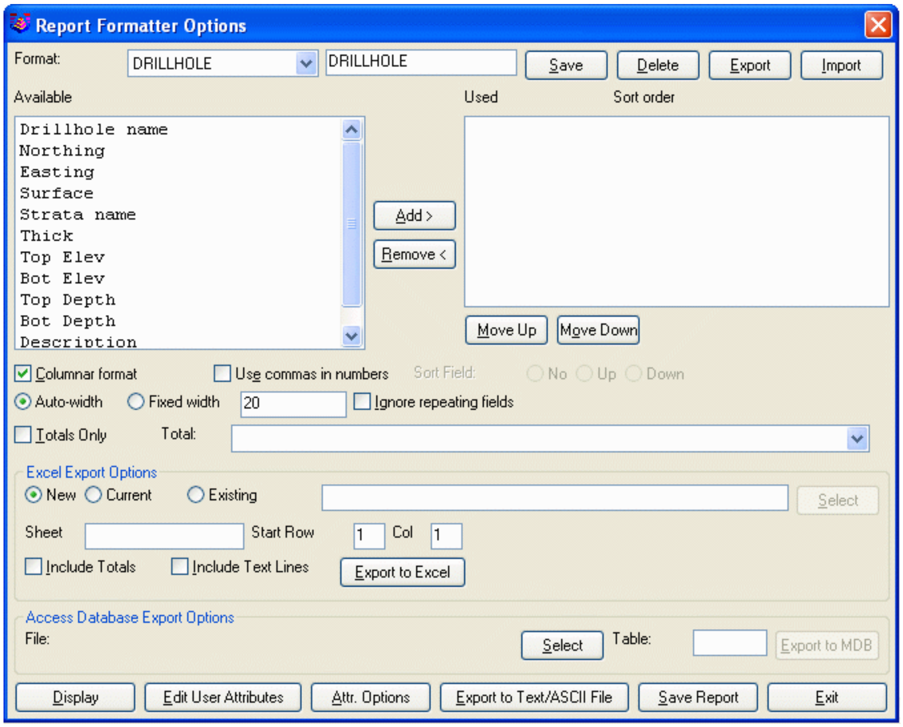
Select objects:

Add Page break between drillholes [Yes/<No>]?
Report Strata depth to [Top/<Bottom>]?
Report Strata elevation of [Top/<Bottom>]?
The report is then displayed accordingly.



Custom Drillhole Report

This function allows you to customize your report format.



Prompts:

Command: tk_chreport2
Select the Drillholes for report.
Select objects: Specify opposite corner: 271 found
262 were filtered out.

Keyboard Command: tk_chreport, tk_chreport2
Prerequisite: drillholes

Make Strata Surface

This command generates multiple strata surfaces based on strata definitions and placements of drillholes. Strata surfaces are generated at the bottom of each strata. These strata surfaces can then be used in other TakeOff commands like **Calculate Total Volumes**. They can be viewed on screen, through the command **Draw Strata Surface**.

Note: By observing the command line, one can see the status of each strata surface generation.

Keyboard Command: tk_chgrid

Prerequisite: Define Drillhole/Strata Settings, Place Drillhole

Clear Strata Surface

This command clears the strata surfaces previously generated with **Make Strata Surface**. This removes the strata surfaces from processing in other takeoff commands.

Note: This command will not remove the surface from the screen view. You must use the command **Erase Strata Surface** to remove them from view.

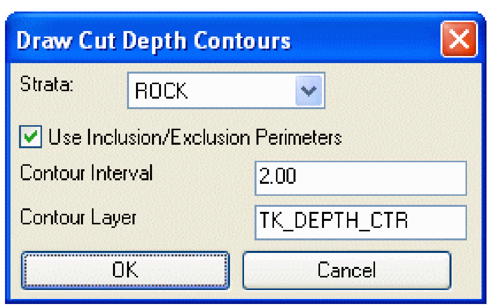
Keyboard Command: tk_chclear
Prerequisite: Make Strata Surface

Draw Strata Cut Depth Contours

This command will draw the **Strata Cut Depth Contours**. This command creates contours for the cut depth between the design surface and strata.

You must have created Strata Surfaces through the **Make Strata Surface** command.

Then select **Draw Strata Cut Depth Contours** from the **Drillhole menu**. You will be prompted to select the Strata from the dialog box below.



You can assign a contour interval and contour layer for the contours to be drawn. If **Use Inclusion/Exclusion Perimeters** is checked on you will be prompted for an Inclusion polyline and a Exclusion polyline if needed, otherwise the drawing's Boundary linework will be used.

Keyboard Command: tk_chdepth
Prerequisite: Make Strata Surface

Erase Strata Cut Depth Contours

This command will erase the Strata Cut Depth Contours from the screen display.

Keyboard Command: tk_chdepth2
Prerequisite: Strata Cut Depth Contours

Draw Strata Cut Color Map

This command will generate a map of areas where the design surface cuts into the selected strata.

Prompts

Select point for color legend: **-** Use your pointing device to select the top left corner of where you want the cut color legend to be displayed.

Legend size <**10.0**>**:** Screen display size.
Label all zones or summary [All/<**Summary**>**]?** This pertains to the number of elevation labels on the legend.

Keyboard Command: tk_chmap
Prerequisite: Make Strata Surface

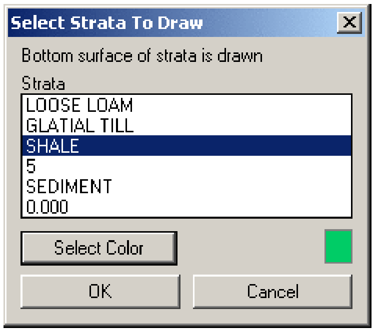
Erase Strata Cut Color Map

This command will erase all Strata Cut Color information from the screen display.

Keyboard Command: tk_chmap2
Prerequisite: Draw Strata Cut Color Map

Draw Strata Surface

This command will display the selected strata surfaces as 3D faces. The bottom elevation of the strata is drawn.



A color can be selected to distinguish each strata.

Keyboard Command: tk_chplot
Prerequisite: Make Strata Surface

Erase Strata Surface

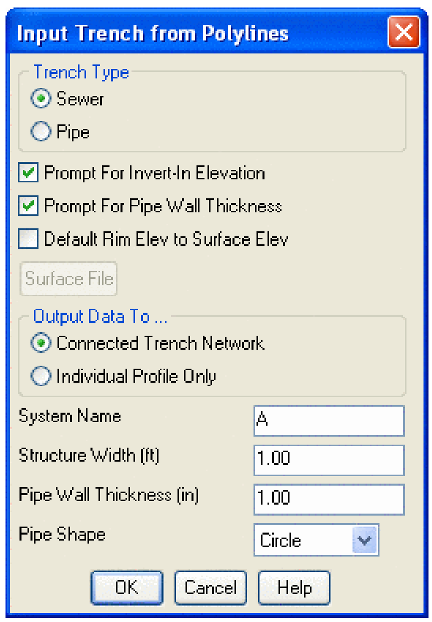
This command will erase all strata surface 3D faces from the screen display.

Keyboard Command: tk_chplot2
Prerequisite: Draw Strata Surface

Trench Menu 10

Input Trench From Polyline

This command allows you to input a trench sewer network structure from polylines. It first prompts you the **Input Trench from Polyline Dialog** where you specify the Trench Type, Trench System, and the System Name. The Individual Profile option lets you input one trench reach at a time and save its information to a profile (.pro). The Connected Network option lets you input all the trench polylines on the drawing, merge them into a trench network structure and save the whole structure to a .sew file. For trenching or utilities without Invert-Ins, uncheck Prompt For Invert-In Elevation. If you want to set the Rim Elevation to any surface elevations, check on Default Rim Elev to Surface Elev and then use the Surface Button to select the desired .tin or .flt surface file. Prompt For Pipe Wall Thickness allows you to enter in the pipe thickness that will be used in calculating backfill quantities in two prompts: 1) the interior pipe size and, 2) the thickness of the pipe wall. If this is check off, the value in Pipe Wall Thickness will be automatically added to the Pipe Size for backfill volumes. You can also enter in a Structure Width to be considered in the Cut volumes. Pipe Shape determines the prompting so that you can create Circular, Elliptical, or Rectangular pipe. Click OK to start inputting trench structure.



Pick a polyline that represents a trench reach on your drawing. Takeoff extracts the coordinates of all the vertices of the polyline and prompts you for the starting station number. Takeoff would computer the station values based on the starting station number. Next, you are prompted to enter the Manhole ID (Sewer Trench) or Station ID (Pipe Trench), Invert Elevation, Manhole Elevation (Sewer Trench), and Pipe Size or Pipe Group for every station. You can either enter the values manually or select the texts that represent these values on the drawing. When you finish inputting a polyline, the command would ask you for a profile name to store the profile data if you are doing Individual Profile; otherwise the command would ask you to pick next polyline that is in the same trench network.

Prompts:

Pick a polyline that represents a trench reach: *pick a polyline on your drawing*
Starting Station of trench reach <0.0>: *press Enter to accept 0.0 as the starting station or enter a value*

For station 0.00 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*
For station 270.22 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*
Undo/Select/Group/<Enter Pipe Size <0.0000>>: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually or select Group to enter in a Pipe Group*
For station 425.02 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *elect the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*
Undo/Select/Group/<Enter Pipe Size <0.0000>>: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually or select Group to enter in a Pipe Group*
For station 649.73 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *elect the invert elevation text on the drawing or enter Enter on the keyboard to enter the invert elevation value manually*
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manhole elevation value manually*
Undo/Select/Group/<Enter Pipe Size <0.0000>>: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually or select Group to enter in a Pipe Group*
Another Polyline [<Yes>/No]? *enter Yes to input another trench reach from a polyline or enter No to finish*

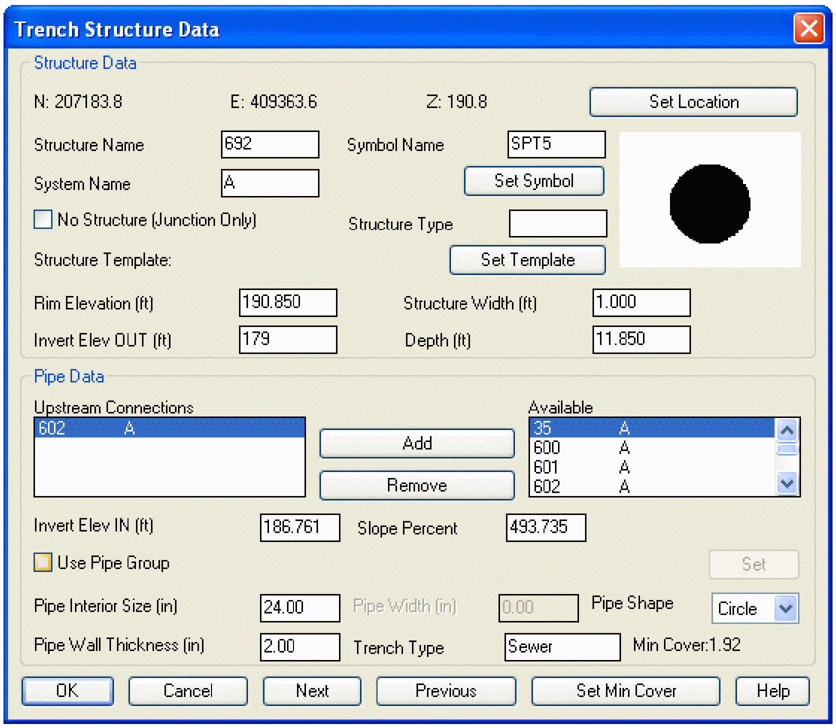
At the end of the command, a file opening dialog would be prompted to you to specify a .sew file name to store the trench network structure.

Prerequisite: A drawing with one or more polylines that represent the trench structure.
Keyboard Command: pline_trench

Create Trench Network Structure

This command allows you to create or modify a trench network structure on a drawing. Before you are able to locate the trench structure, the drawing has to be open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. You

can locate the trench structure by one of three methods: picking points on the drawing, entering the point number, or specifying the station and offset of a centerline. If you use centerline method, you need to specify a centerline file. After you locate a point on the drawing, you are prompted the **Sewer Structure Data Dialog** for entering the sewer structure information, such as Structure Name, System Name, Symbol Name, and Elevations. Take a look at the list of the trench points that have been defined. If there is any point that is connected upstream to the current point, you add it to the Upstream Connections list. The Invert Elevation and the Pipe Size fields will be filled with the information of the upstream point. Use Pipe Group allows you to set multiple pipes for the trench run by using a existing or new Pipe Group. Click OK to finish entering the trench structure data. The command will repeatedly ask you to pick a structure point until you hit **Enter** to finish. The trench network structure data is saved in a .sew file.



Prompts

By Pick:

Locate by pick point, point number or station-offset [<Pick>/Number/CL]? *press Enter to do Pick point*

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Pick structure location: *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*

Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *press Enter to finish*

By station-offset of CL:

Locate by pick point, point number or station-offset [<Pick>/Number/CL]? CL (enter CL to do locating trench structure by station-offset of a centerline)
Specify a centerline file.

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Structure Station: 0 *(enter the station number on the centerline)*
Structure Offset: 200 *(enter the offset from the centerline)*
Sewer Structure Data Dialog: *enter trench structure information*
Structure Station (Enter to end): 100 *(enter the station number on the centerline)*
Structure Offset: 200 *(enter the offset from the centerline)*
Sewer Structure Data Dialog: *enter trench structure information*
Structure Station (Enter to end): *press Enter to finish*

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface.
Keyboard Command: locate_trench

Edit Trench Network Structure

This command edits the existing trench structure data on the drawing. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing. If there is no such structure in the .sew file, you would get a error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command will restore the trench structure data from the corresponding .sew file and display it on the Sewer Structure Data Dialog for editing. Click OK to confirm your modification. You are prompted to edit another structure point until you press Enter to finish. All modifications are saved in the .sew file.
Set Location will return you to the plain view and prompt you for a new location for the structure by either typing in the coordinates or picking on the screen. In the dialog you can change the Structure Name, Symbol, Width, Depth, and Type. Setting a Structure Template will allow you to set the dimensions of the Structure with a .tch file. See Input-Edit Trench Template for details on creating a .tch file. Here you can also manage how the Structure is connect to other Structures. Under Upstream Connections you will see the Structure(s) currently connected to upstream and a list of available Structures on the right. Pick Add to connect to a Structure you have selected under Available, and Remove to disconnect to any selected Structures. Other options are to edit the Rim Elevation, Invert In and Out, as well as Pipe information between your Structure and the highlighted Upstream Connection. Use Pipe Group allows you to set multiple pipes for the trench run by using a existing or new Pipe Group. Min Cover shows you the depth between the Design Surface and top of pipe. Set

Min Cover will adjust your Invert In and Out elevations so that you have at least the value you enter as the Min Cover.

Prompts

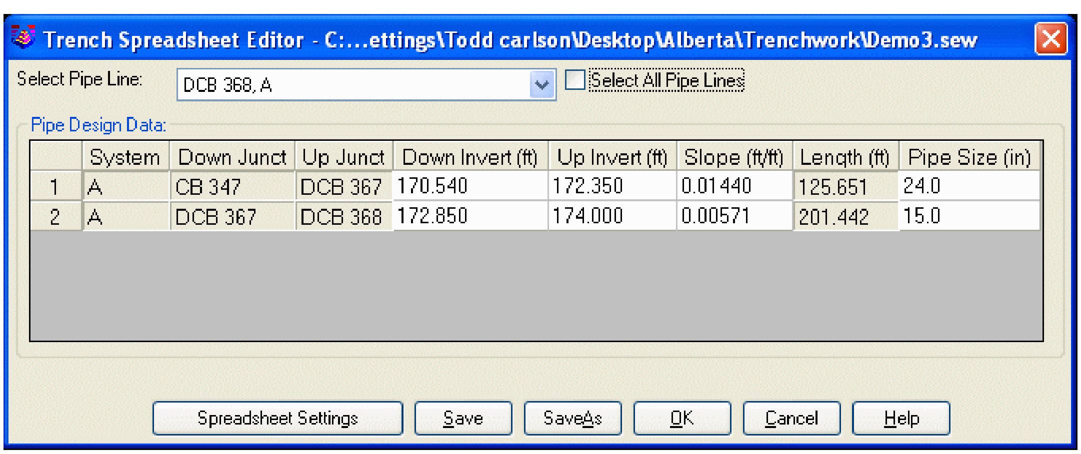
Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Select sewer structure to edit: pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): press Enter to finish

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.
Keyboard Command: edit_trench

Trench Spreadsheet Editor

This command allows you to view and edit existing trench network data in spreadsheet form. Upon running the command, the program will open the .sew file associated with the drawing, or if one has not been established, you will be prompted to select one.



The Trench Spreadsheet Editor allows you to select the Pipe Line you want to edit, or view all the Pipe Lines at once by checking on "Select All Pipe Lines". After selecting a Pipe Line, each segment of the Pipe Line will be displayed as: the downstream connection (Down Junct), upstream connection (Up Junct), the invert in of the downstream manhole (Down Invert), the invert out of the upstream manhole (Up Invert), and the Slope, Length and Pipe Size between the two. Any value between two manholes can be edited except for the Length. Spreadsheet Settings allows you to choose what elements of a segment are displayed.

Click OK to confirm your modification. All modifications are saved in the .sew file.

Prerequisite: Sewer Network File
Keyboard Command: edit_trench2

Remove Trench Network Structure

This command removes the existing trench structure data. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing or to select from a List of your Sewer Structures. If there is no such structure in the file, you will get a error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command removes the structure from both the drawing and the .sew file immediately. You are prompted to remove another structure point until you press **Enter** to finish. The removed trench structure points would no longer be found in the .sew file.

Prompts

Select structures to erase by screen pick or name list [<Pick>/List]? Pick to choose from the screen, or List to choose from the below dialog.

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: remove_trench

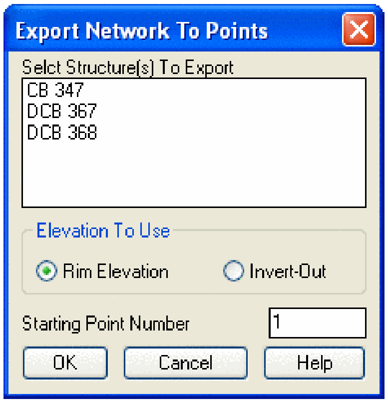
Find Trench Network Structure

This command will center the screen and draw an arrow to the structure you specify.

Prerequisite: a Trench Network
Keyboard Command: findswr

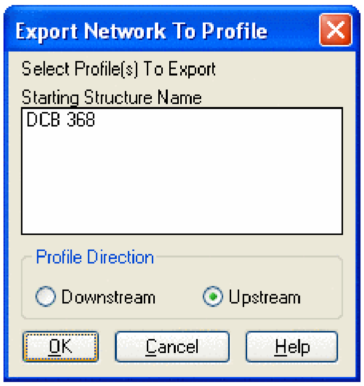
Export Trench Network Data

Export to Points



This command will add points at your trench structures and add them into your coordinate file by either the Rim Elevation or the Invert-Out.

Export to Profiles



This command will create a profile file (.pro) of your trench either going Upstream or Downstream. The (.pro) file can then be drawn under Roads->Draw Profile.

Prerequisite: a Trench Network
Keyboard Command: swr2pts, swr2pro

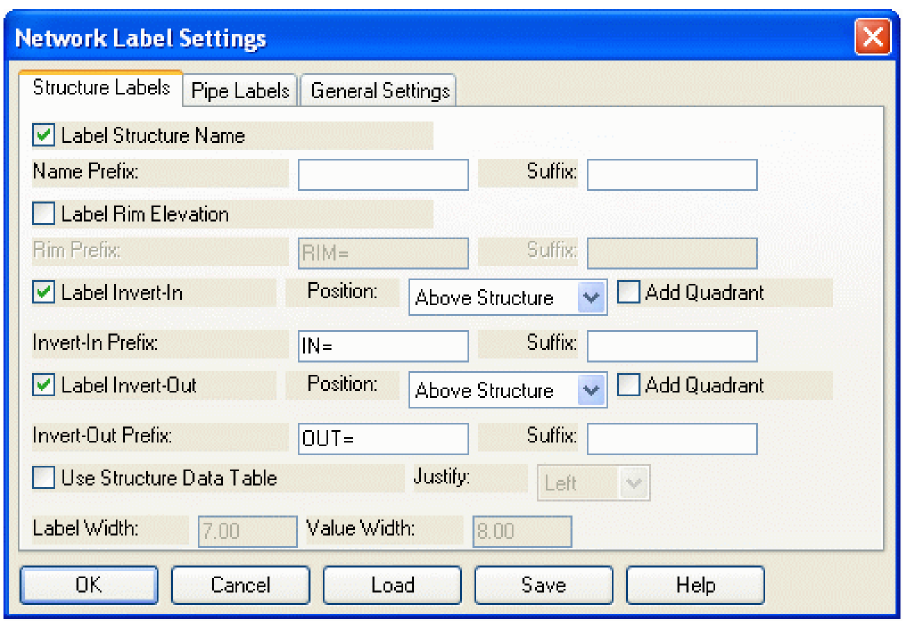
Trench Network File Backup

Save Trench Network File saves your trench network as a (.sew) file. *Load Trench Network File* loads a previously saved (.sew) file.

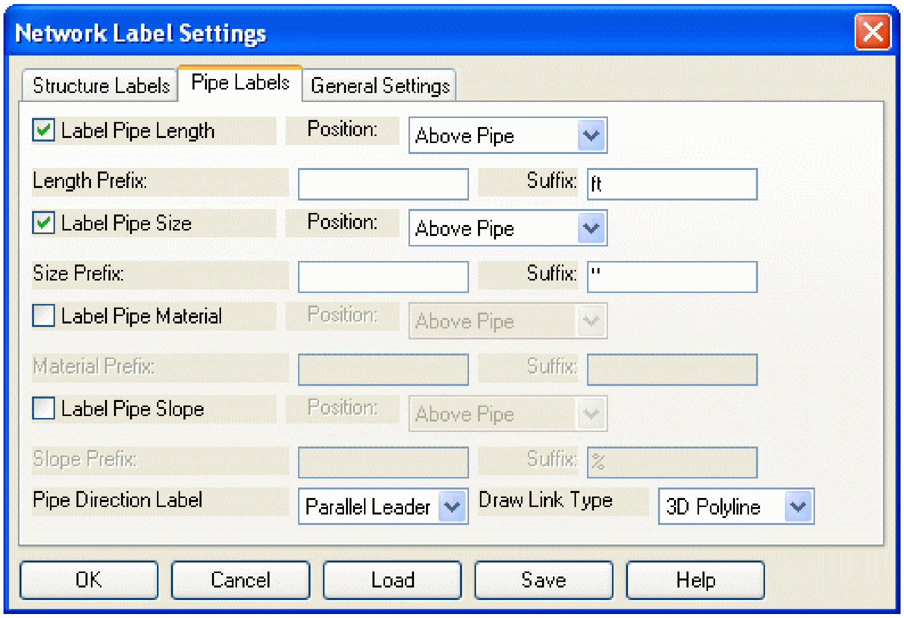
Prerequisite: none
Keyboard Command: save_trench, load_trench

Plain View Label Settings

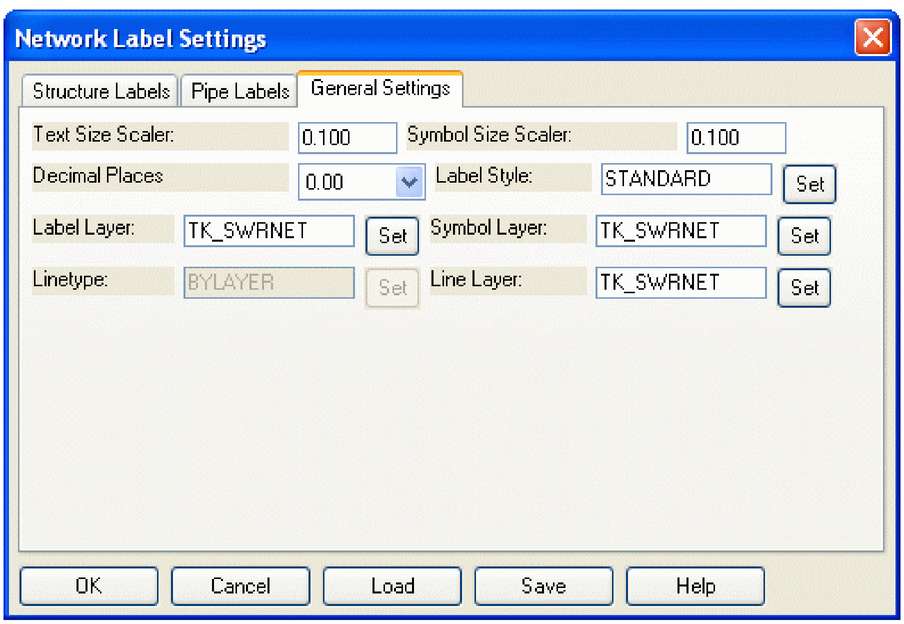
This command allows you to set the labeling for your structures and piping. The below dialog box gives you the option to display the Structure Name, the Rim Elevation, the Invert-In, and Invert-Out. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. The Use Structure Data Table will create linework around each Structure's labeling.



This below dialog box gives you the option to display the Length, Size, Material, and Slope for you Piping. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. To specify to which structure the label is meant for, select Arrow On Pipe, Parallel Leader, or None. You can also set the type of linework to draw.



In this dialog you can set the properties for your Symbol and Linework as well as the decimal places to report.



Prerequisite: a trench network
Keyboard Command: swrsetup

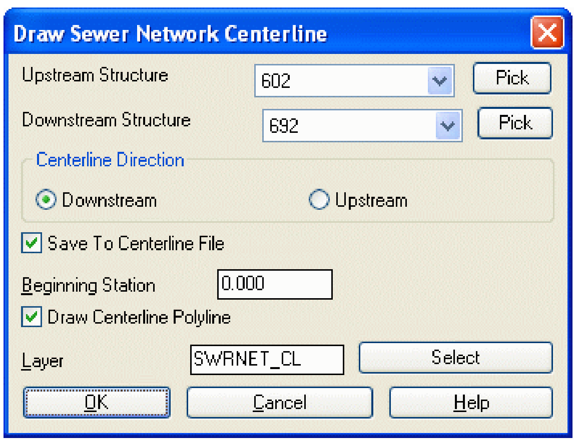
Draw Trench Network - Plan

This command draws a trench network structure on the screen, based on the Plan View Label Setting command and the trench network structure data in the .sew file whose name is as same as the drawing name. If Takeoff couldn't find such file in the same directory where the drawing locates, nothing would be drawn on the screen.

Prerequisite: A open drawing
Keyboard Command: plan_trench

Draw Trench Network Centerline

This command allows you to draw a branch of the trench network structure as a centerline. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get a error message like "Error: no data in sewer network file". The command first prompts you the **Draw Sewer Network Centerline Dialog**. Select the Upstream and Downstream Structure for the centerline you are about to create. The Centerline Direction determines from which structure the polyline is drawn. You can also choose to save the centerline data to a .cl file with the option of entering in the Beginning Station. In this dialog is the ability to set the Layer name as well. Click OK to draw.

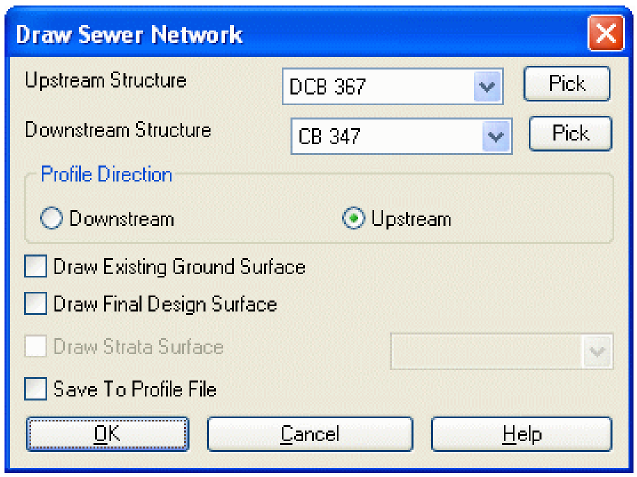


Prompts

Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.
Keyboard Command: drwswrcl

Draw Trench Network - Profile

This command allows you to draw a branch of the trench network structure as a sewer/pipe profile. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get a error message like "Error: no data in sewer network file". The command first prompts you the **Draw Sewer Network Dialog**. Select the Upstream and Downstream Struct that you want to draw. If you want to draw the existing and final design surface, as well as Strata Surfaces, toggle on Draw Existing Ground Surface, Draw Final Design Surface, and Draw Strata Surfaces options. If your profile is from upstream to downstream, then select the Profile Direction as Downstream, otherwise Upstream. You can also choose to save the profile data to a profile file. Click OK to draw.



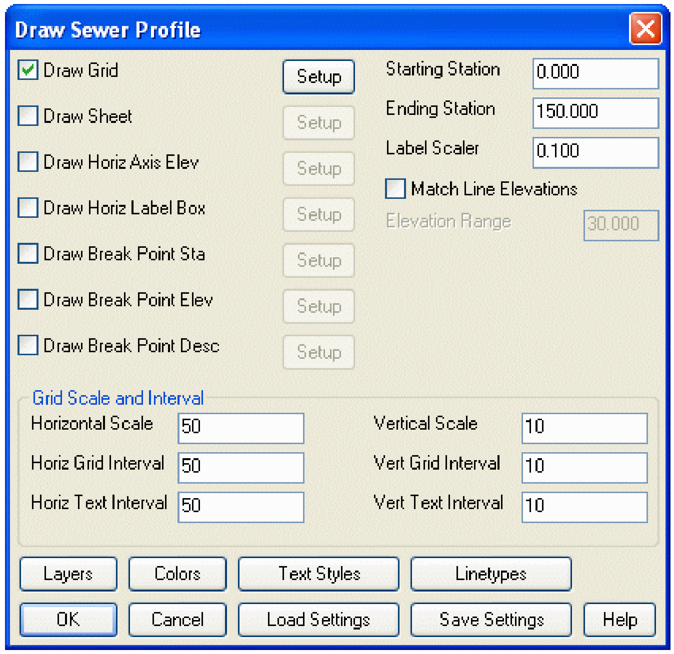
Prompts

Loading edges...
Loaded 41041 points and 122901 edges
Created 81859 triangles

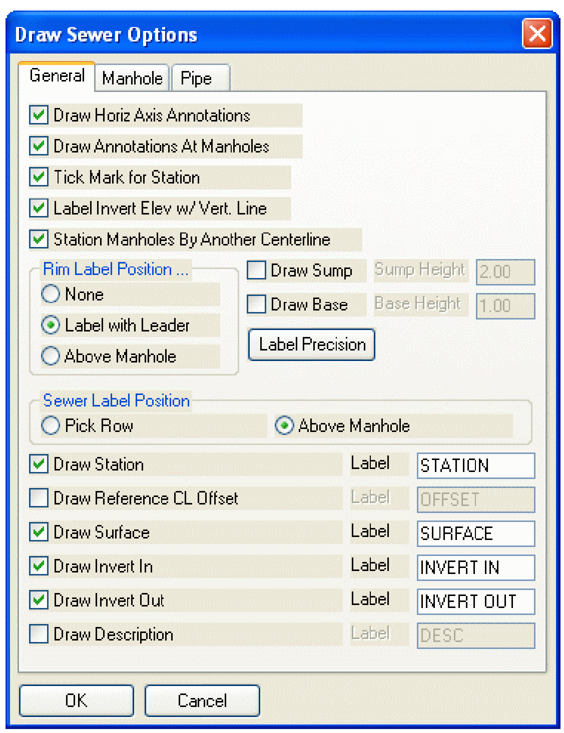
Loading edges...
Loaded 5487 points and 16332 edges
Created 10846 triangles

Initializing Draw Profile command ...
Draw Sewer Profile Dialog

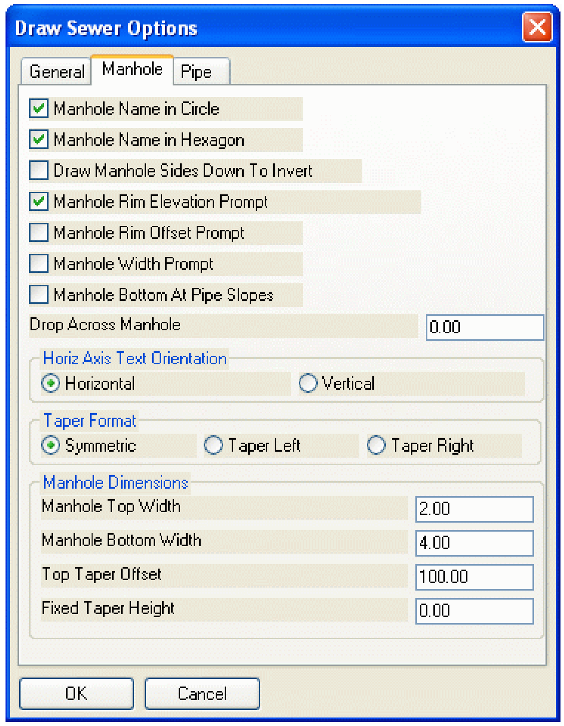
Enter drawing parameters such as Grid scale, text scaler, starting and ending stations etc. for drawing the sewer profile.



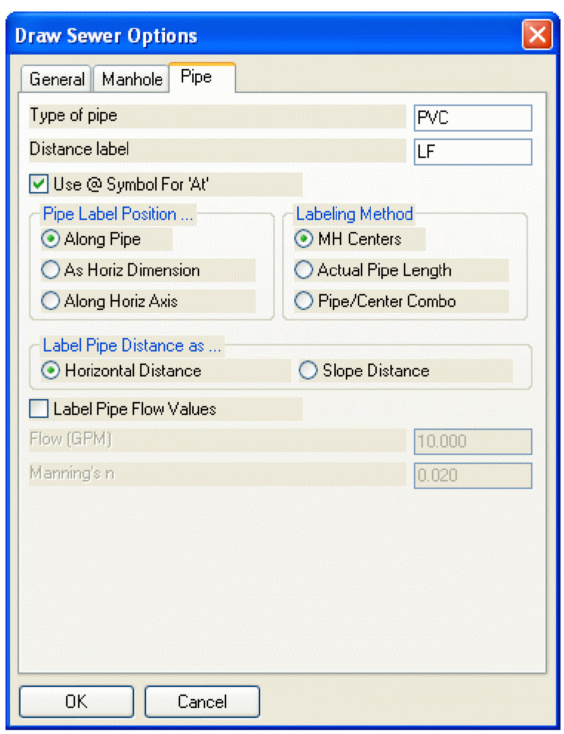
Enter general sewer profile settings such as elevations (Rim, Invert-In, Invert-Out) to draw and label.



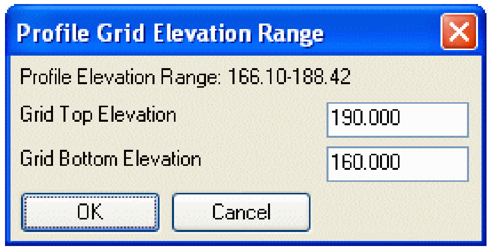
Use the Manhole tab to define what manhole information is labeled in your trench profile.



Use the Pipe tab to define what piping information is labeled in your trench profile.



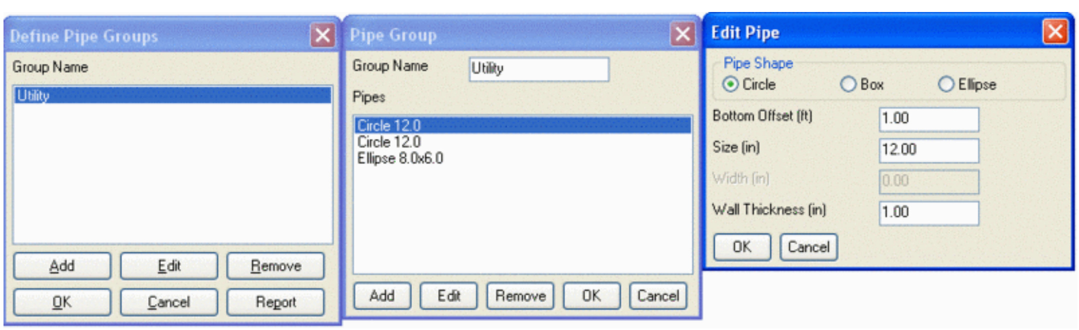
The command will find the elevation range of your profile and display it at the top of this dialog. Here you can set the elevation top and bottom of the profile's grid.



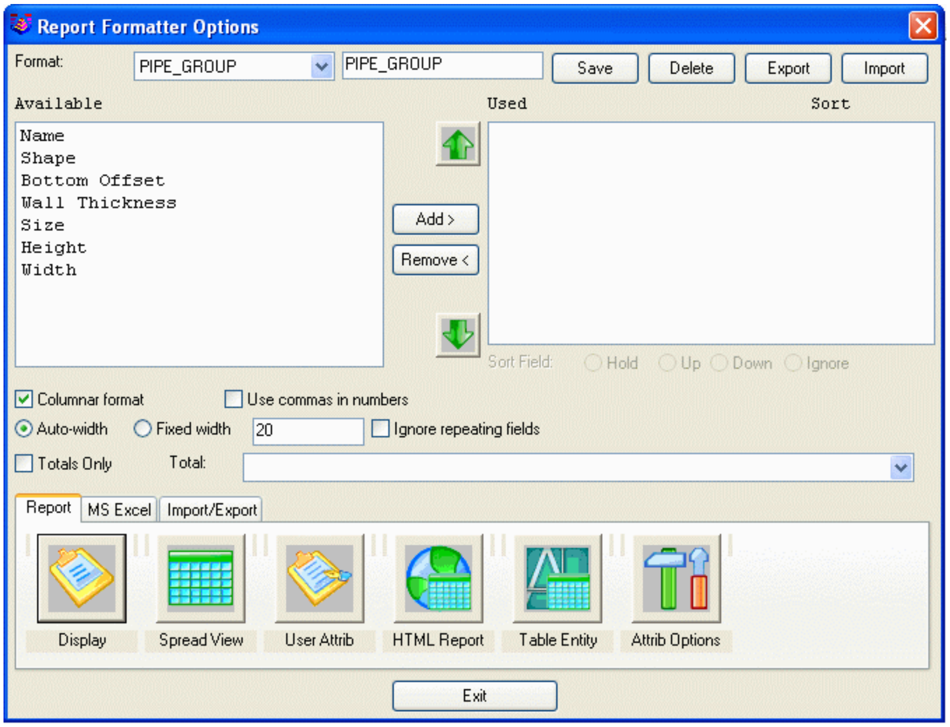
Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.
Keyboard Command: profile_trench

Define Pipe Groups

Pipe Groups allow users to define multiple pipes inside a single trench. Pipe Groups can be applied to a Trench Run during the creation of a Trench Network or after the fact with Edit Trench Structure. In the Define Pipe Groups dialog you can Add, Remove, Edit, or Report different Pipe Groups.



Click Add and a blank Pipe Group dialog will appear. Here you can enter in a Name for the Group and Add different types of pipes into the Group. Clicking Edit or Add in the Point Group dialog will bring up the Edit Pipe dialog. Here you can specify the Pipe Shape, Bottom Offset (from the bottom of the Trench), the Size, Width (when needed), and the Wall Thickness of the pipe.
Report in the Define Pipe Groups dialog will bring up the below dialog:

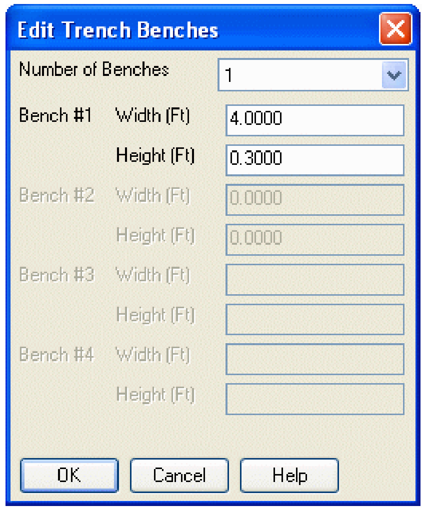


From this dialog you can pick on what you want to report and view it or export it into Excel.

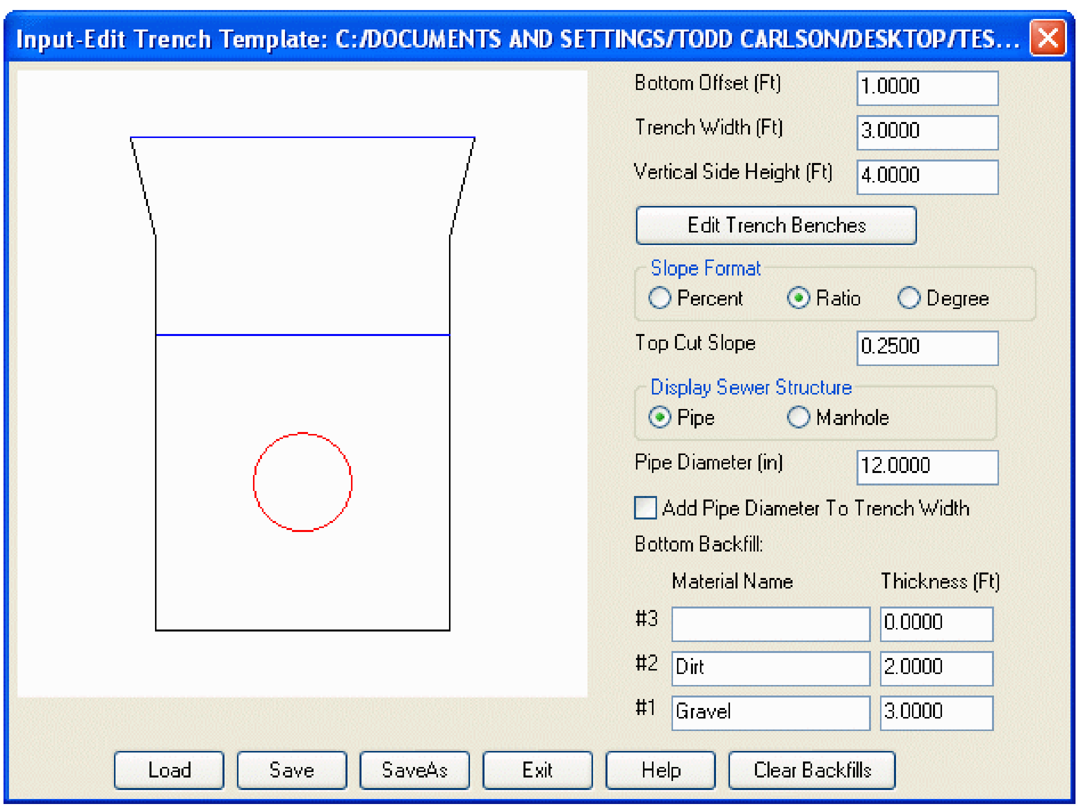
Prerequisite: none
Keyboard Command: define_pipe_grp

Input-Edit Trench Template

This command lets you create a new trench template or modify an existing trench template. It prompts you the **Input-Edit Trench Template Dialog**. If you are modifying a trench template, click the **Load** button on the dialog to open a trench template file and display the template data on the dialog. Enter the dimensions of the trench: bottom offset, trench width and vertical side height. The Edit Trench Benches button will bring up the below dialog, and allows you to enter in up to four benches into your trench.



There are three methods for entering the cut slope, Percent, Ratio and Degree. Choose one of the methods and enter the slope value. Display Sewer Structure allows you to see your pipe or manhole as part of the trench. This is for display purposes only, calculations will be drawn from the pipe size you set in the Trench Network Structure commands. Add Pipe Diameter To Trench Width will increase the size of your trench by the diameter of your different pipe sizes. There are three trench bottom backfill layers that can be defined. Enter the layer label in the material name field, the depth of the layer in the thickness field. Click Save or SaveAs to save the template information in a .tch file, and Click Exit to quit this command.



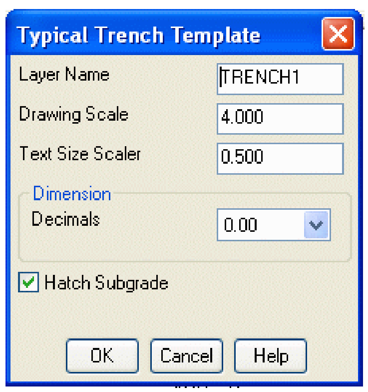
Prompts:

Input-Edit Trench Template Dialog
Enter the dimensions of the trench template, save the information to a template file (.tch).

Prerequisite: None
Keyboard Command: make_trench_tpl

Draw Typical Trench Template

This command draws a trench template on the screen. After you select a trench template file (.tch) to draw, a **Typical Trench Template Dialog** is prompted for entering the layer name, drawing scale, text size scaler and selecting how many decimal points you want. You can also hatch the backfill on the drawing. Click OK to draw the template at the position that you pick on the screen.



Prompts

Pick position to draw template: *pick a position on the screen*

Prerequisite: None.
Keyboard Command: draw_trench_tpl

Trench Subgrade Areas

The purposes of Trench Subgrades is to assign a different type of trench template when a trench passes under a road, building pad, etc.

Set Trench Subgrade Polylines

Choose a closed polyline that defines the area that you want a different trench template for, ie a building pad polyline.

Prerequisite: None.
Keyboard Command: tag_trench_subgrade

Clear Trench Subgrade Polylines

This command untags selected polylines for trench subgrade use.

Prerequisite: Trench Subgrade Polylines
Keyboard Command: untag_trench_subgrade

Hatch Trench Subgrade Area

This command hatches trench subgrades for easy viewing.

Prerequisite: Trench Subgrade Polylines
Keyboard Command: hatch_trench_subgrade

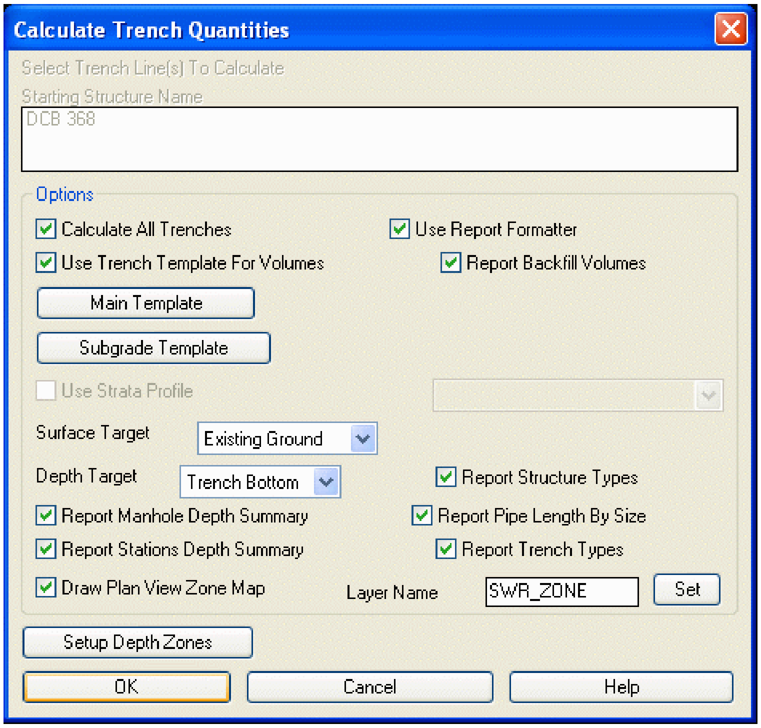
Erase Trench Subgrade Hatch

This removes previous made trench hatching.

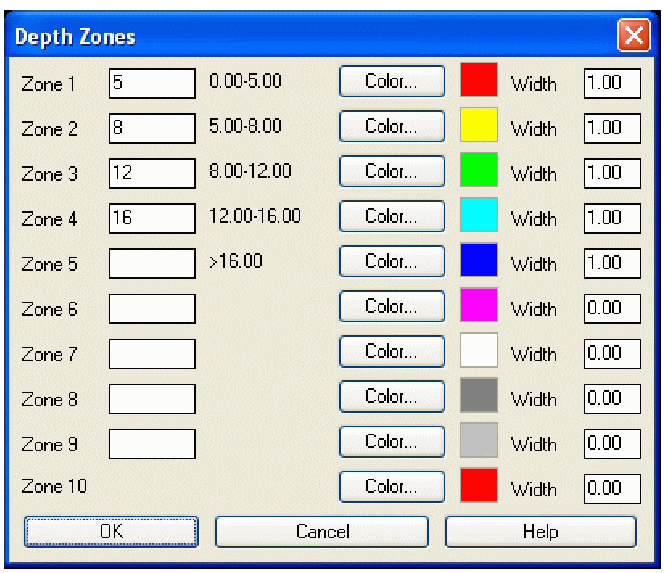
Prerequisite: Trench Subgrade Hatching
Keyboard Command: erase_trench_subgrade

Trench Network Quantities

This command calculates the trench volumes. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get a error message like "Error: no data in sewer network file".
The command loads the trench network data and split them into individual trench lines and display them on the **Calculate Trench Quantities Dialog**. You can choose to calculate the trench volume of one trench line or several trench lines at a time. If you select **Use Trench Template for Volumes**, you need to **Set Trench Main Template** and **Subgrade Template** if you have one. Surface Target determines the Surface that bottom of the trench is compared to, either: the Existing Surface, the Design, the Existing and Design to minimize cut, or simply to the Rim Elevations (no surface required). Trench Depths can be reported by either the bottom of the trench or bottom of the pipe by using the Depth Target pull-down. If you have Strata Surfaces defined then the program can calculate cut volumes for a strata you select.

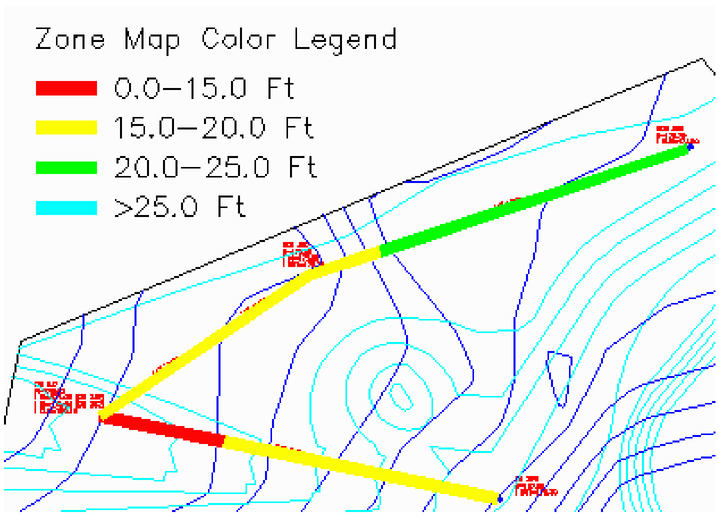


Setup Depth Zones will report you your Trench depth zones by stations along the trench network. You can also color the trench in the drawing by defined zones. Click OK to compute the template volumes. Backfill quantities take into account pipe size. A report would be shown after the calculation.



Prompts

Loading edges...
Loaded 41041 points and 122901 edges
Created 81859 triangles
Loading edges...
Loaded 5487 points and 16332 edges
Created 10846 triangles
Trench Quantities Report Window
Draw zone map color legend on the screen [Yes/<No>]? *y for Yes*
Pick a point for color legend: *pick a point away from site*
Legend size <10.00>:*Press <Enter> for the default*

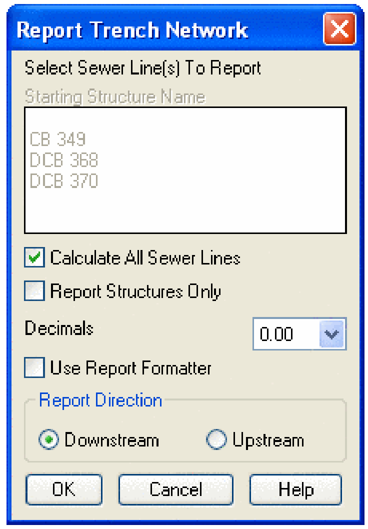


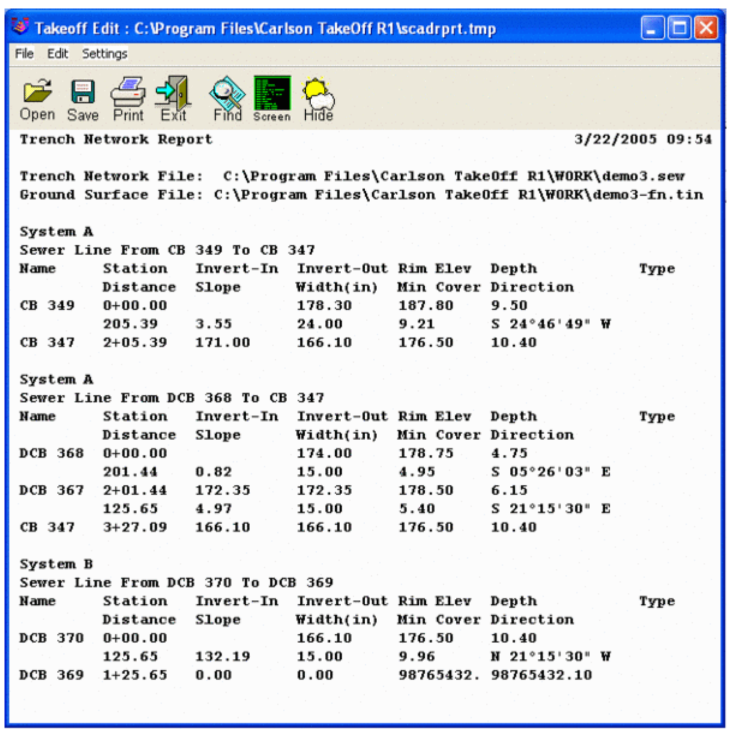
Prerequisite:
Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: calc_trench

Report Trench Network

This command will a report the Name, Station Distance, Invert-In Slope, Invert-Out Width, the Rim Elevation, Trench Type, Manhole Depth, and the Area Direction for the selected Trench. You may also choose to report the Trench Network from Downstream or Upstream, or just the Structures.






```
Trench Network Report                                              3/22/2005 09:54

Trench Network File:   C:\Program Files\Carlson TakeOff R1\WORK\demo3.sew
Ground Surface File: C:\Program Files\Carlson TakeOff R1\WORK\demo3-fn.tin

System A
Sewer Line From CB 349 To CB 347
Name      Station    Invert-In   Invert-Out Rim Elev  Depth            Type
          Distance   Slope       Width(in)  Min Cover Direction
CB 349    0+00.00                178.30     187.80    9.50
          205.39     3.55        24.00      9.21      S 24°46'49" W
CB 347    2+05.39    171.00      166.10     176.50    10.40

System A
Sewer Line From DCB 368 To CB 347
Name      Station    Invert-In   Invert-Out Rim Elev  Depth            Type
          Distance   Slope       Width(in)  Min Cover Direction
DCB 368   0+00.00                174.00     178.75    4.75
          201.44     0.82        15.00      4.95      S 05°26'03" E
DCB 367   2+01.44    172.35      172.35     178.50    6.15
          125.65     4.97        15.00      5.40      S 21°15'30" E
CB 347    3+27.09    166.10      166.10     176.50    10.40

System B
Sewer Line From DCB 370 To DCB 369
Name      Station    Invert-In   Invert-Out Rim Elev  Depth            Type
          Distance   Slope       Width(in)  Min Cover Direction
DCB 370   0+00.00                166.10     176.50    10.40
          125.65     132.19      15.00      9.96      N 21°15'30" W
DCB 369   1+25.65    0.00        0.00       98765432. 98765432.10
```

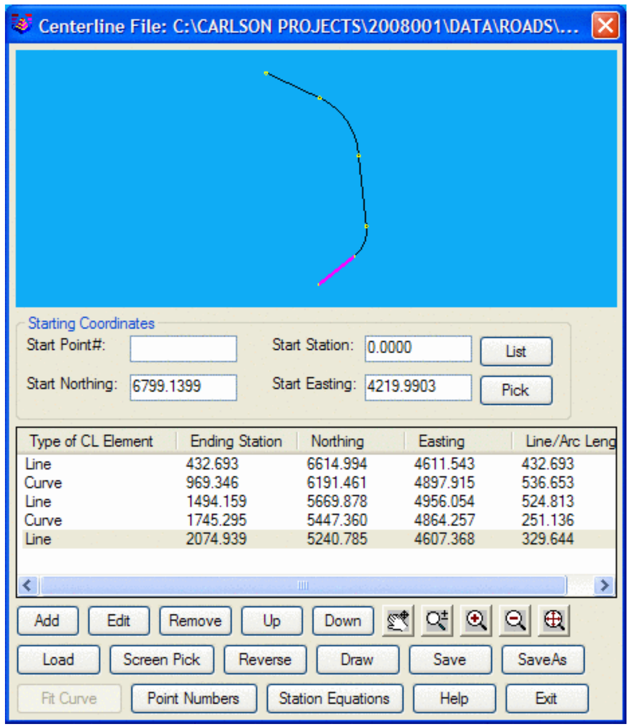
Prerequisite: a sewer line
Keyboard Command: reportswr

Roads Menu 11

Input-Edit Centerline File

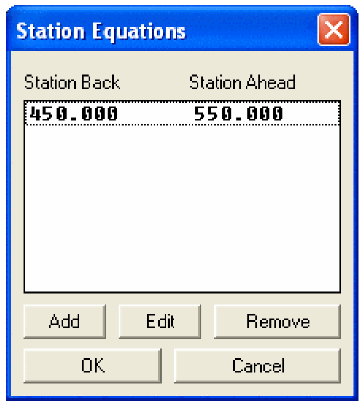
This command can be used to input a new centerline or edit an existing centerline (.CL) file. It is a dialog-based alternative to *Design Centerline* and has the advantage of accepting whatever information you have on your centerlines (coordinates, stationing, length of tangents and arcs, etc.). For creating a new centerline, it is ideal for entering data straight from highway design plans. For editing, this command allows you to change any of the geometric properties of any of the elements of the centerline (lines, curves, spiral-only and symmetrical spiral-curve-spiral elements), including the starting coordinates and station.

Starting this command launches the Centerline Input-Edit main dialog box. To edit an existing Centerline, you can either pick the Load button and pick the .CL file, or pick the Screen Pick button and pick the polyline in the drawing that represents the Centerline. The Centerline is then displayed in the graphics window of the dialog box. The highlighted segment in the text window is also highlighted in the graphics window.



Up/Down: Moves elements in the table Up and Down in the list. For example, if this centerline ended with a tangential line from the last curve, then was followed by a non-tangential line at 45 degrees NE, moving the last element up would create a line at 45 degrees after the curve (non-tangential), and the formerly tangential line will remain tangential and therefore continue at NE 45 degrees.
Draw: This button draws the centerline in the drawing on the specified layer.
Drag Action (Zoom and Pan): In the graphics window, hold the left mouse button down and move mouse to Pan, roll the wheel to Zoom.
Load: Loads an existing centerline (.CL) file for review or editing. After loading a centerline, the listbox in the dialog shows a list of all the elements in the centerline, identifying them as either a line, curve, spiral only or full spiral-curve-spiral element and reporting the ending station, northing and easting of the element.
Add: Adds a new element after the highlighted element. Prompts you for the type of the element to be added, Line, Curve, Spiral-Only or Spiral-Curve-Spiral.
Edit: Allows you to edit the highlighted segment.
Remove: Removes the highlighted element from the centerline.
Assign Point Numbers: This will create Carlson points along the elements of the centerline and store them to the

current CRD file. The new points will be numbered in sequence beginning with the first available point number in the CRD file.
Reverse: Reverses direction of Centerline.
Save: Saves the currently loaded centerline to a file, or will prompt you for a name if no name has been set.
SaveAs: Prompts you for a file name for the saved file.
Exit: Exits this routine, prompting to save changes if necessary.
Help: Goes to the Help screen for the Input-Edit Centerline command.
Station Equations: At any number of locations on a centerline, you can set the back station and forward station for the re-stationing of the centerline. The station equation dialog appears below:



If the Station Back is lower than the Station Ahead, then a "gap" is inserted in the centerline, where the stations jump forward. If the Station Ahead is less than the Station Back, then an overlap occurs, where the common station range is repeated.

The dialog for every type of element shows the point ID, the northing, easting and station of the start point of the element. It then allows the user to modify or define the parameters specific to the type of element. The following are some of the things to remember about data entry in the centerline editor. These are valid for lines, curves and spirals.

• Wherever length of the element is to be entered, entering an expression of the type 123.5 - 93.7 would evaluate the difference of the values. This is particularly convenient where only the stations of the start and end points of the element are known.
• When the station is specified, the program takes the length of the element as the difference between the station of the start point of the element and the station specified.
• All bearings should be specified by entering the angle between 0 and 90 degree (in dd.mmss format) and selecting the quadrant.
• When entering the delta angle of a curve, only the absolute value (between 0 and 360 degree) is to be entered. The direction of the curve is to be explicitly set as right or left, the default being left. All angles are entered in (dd.mmss) format.
• Point numbers, when used, access their coordinates in the current .CRD file. If the point number specified has no coordinates stored in the coordinate file, the point number is remembered for that particular location (say the radius point of a curve or the SC point of a spiral). Then, when the .CL file is saved, the program creates points for that location and stores them to the .CRD file with the specified point number.

The dialog for a Line allows the user to specify the line primarily by its length or station and its bearing. The line can also be defined by its end point number or its coordinates. The bearing of a line can be changed if the Tangential to the Previous Element toggle is not checked. By default, any line which follows a curve element is defaulted to be tangential to it. To use a bearing different than that of the previous element, uncheck this toggle and enter the bearing.

The dialog for the Curve allows the user to define the curve primarily by its radius and delta angle or arc length. The other parameters of the curve that can be edited are the bearing of tangent-out and the "Station to", which also defines the arc length. The curve can also be specified by entering the coordinates or point numbers of its end point (PT) and the radius point. Another way to specify the curve would be to enter the chord length or PT point station and chord bearing. If the central PI point and a point on the forward tangent are known, then the curve can be defined by entering both of these points and at least one other property of the curve (like radius, arc length, delta angle). The point on the forward tangent can be any point that defines the tangent out direction including the next PI point. If only the central PI point is known, then the tangent-out can be entered by bearing instead of by forward tangent point. Central PI and forward tangent points are not displayed from the .CL file. They have to be entered by the user and are valid only for that particular edit session; that is, they are not remembered the next time the file is loaded. Curves are assumed to be tangent to the last element unless the Tangential to the Previous Element checkbox is cleared.

The Curve Edit Mode option defines how the curve is accepted in the centerline. If the Hold PC point is checked on, the radius is taken as fixed and the delta angle of the curve is calculated based on some additional parameter. Hence, the extent of the curve is unlimited. However, if the Hold PI points option is checked on, the bearing of tangent-out of the curve is taken as fixed and the radius is calculated based on some other parameter. In this case, the curve is completely restricted within the central PI point and the bearing of tangent out. Hence, when the Hold PI points option is checked on, the above parameters should also be defined to carry out the calculations.

The dialog for the Spiral-Curve-Spiral element allows the user to define the spiral by entering either the various parameters of the spiral (like the angles and lengths) or the coordinates or point numbers of its defining points: the TS (Tangent-to-Spiral), SC (Spiral-to-Curve), Radius point, CS (Curve-to-Spiral), ST (Spiral-to-Tangent) and end point (optional). While defining the spiral by its geometric properties, the program will accept the data even if the information for the simple curve is given with zero spiral lengths. In this method, however, the central PI point of the spiral MUST be specified (that is, it is always in Hold PI Points mode). The tangent out can be defined by entering bearing or by specifying a point on the forward tangent. This forward tangent point can be the next PI coordinates. The direction of the spiral-in and spiral-out elements would be the same as the direction of the simple curve (left or right).

The spiral can be defined by several different parameters and the order that you enter data into the spiral dialog can be important. There are two main sequences for entering data. The method to use depends on the spiral data that you have. The first method is to enter the radius of the simple curve, the spiral in and out lengths, the tangent bearing out and the PI station. The second method is to make a Line segment coming up to the TS (tangent to spiral) point. This Line segment should be added before creating the Spiral element. Then with the Spiral In point set to the TS point, enter the radius of the simple curve, the spiral in and out lengths, the curve direction (left or right) and the arc length of the simple curve. Then the rest of the spiral points will be calculated.

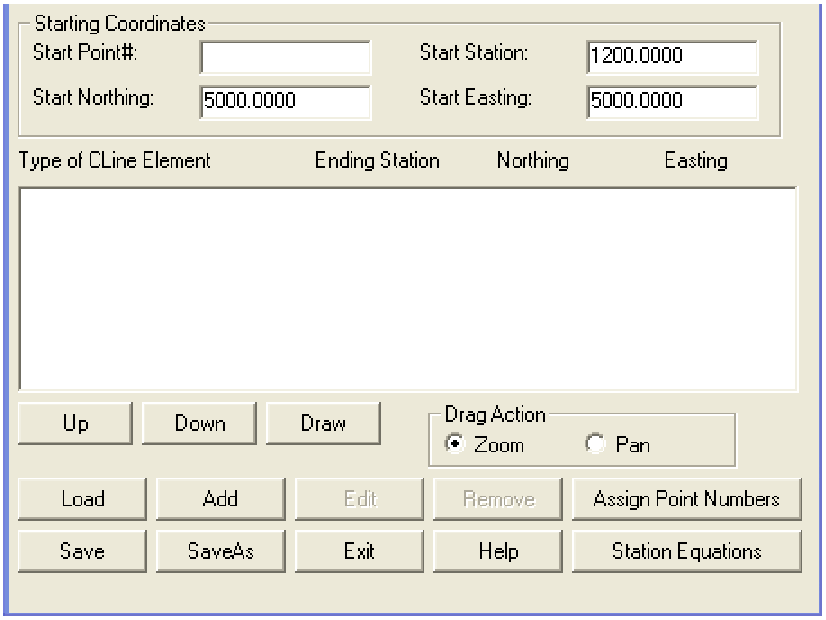
The Spiral Only element allows for flexible transitions from curve to spiral to curve or line to spiral to curve or between any combination of curve and line elements. The Spiral-Curve-Spiral element, for example, can be entered as Line, Spiral Only, Curve, Spiral Only and Line, producing the same results.

Once all the elements of the centerline are defined, the file can be saved and then plotted using the *Draw Centerline File* command.

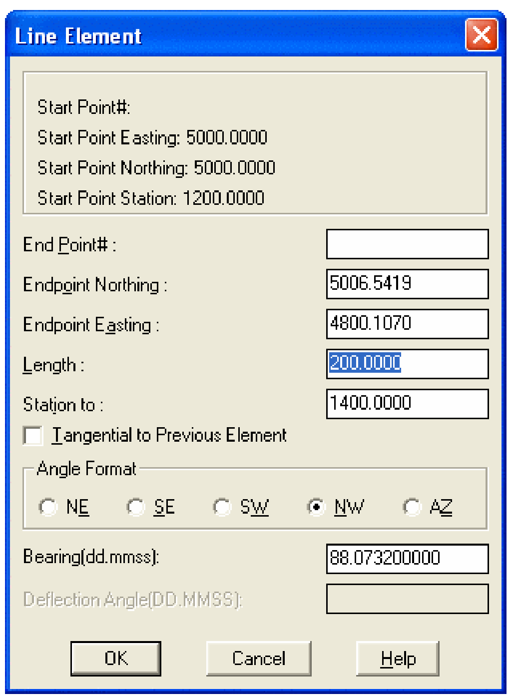
Example

Here is an example of a highway interchange ramp that involves a starting tangent and a spiral curve that goes abruptly into a simple curve and then a final tangent. Here is the starting dialog.

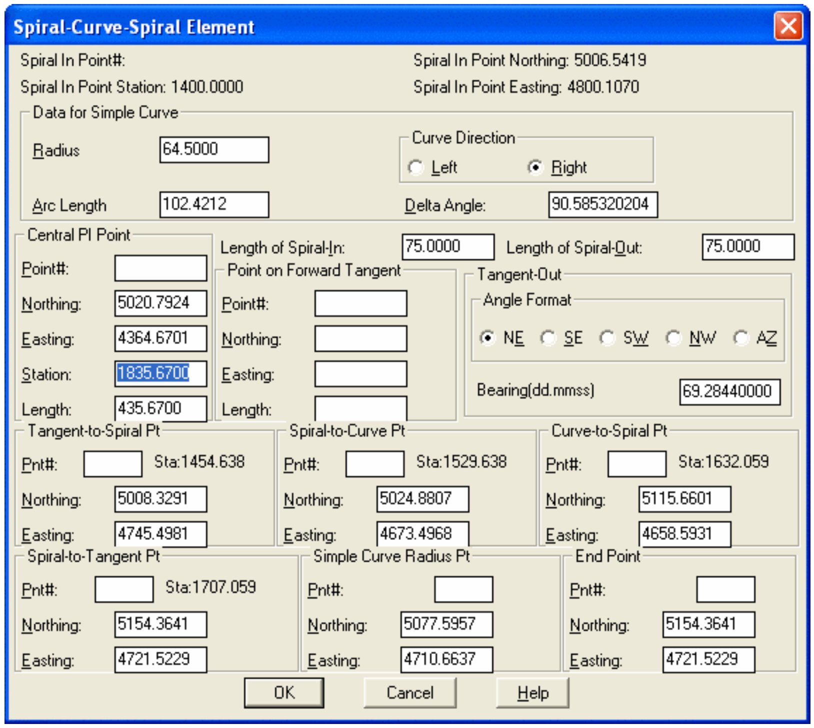
You start by entering a starting Northing and Easting and starting Station. The Start Point# is optional. Then the concept is that you click Add to add each subsequent element (line, curve, spiral-curve-spiral or spiral only).



Line (Tangent) Segment: We want to enter the tangent segment length up to the TS (tangent to spiral). Enter in the length (200.0), bearing (88.0732) and then the bearing quadrant (NW). Since the next spiral-curve-spiral element can be based on a PI station, it is not necessary for this line segment to go up to the TS point. The purpose of this line segment is to establish the tangent-in direction.

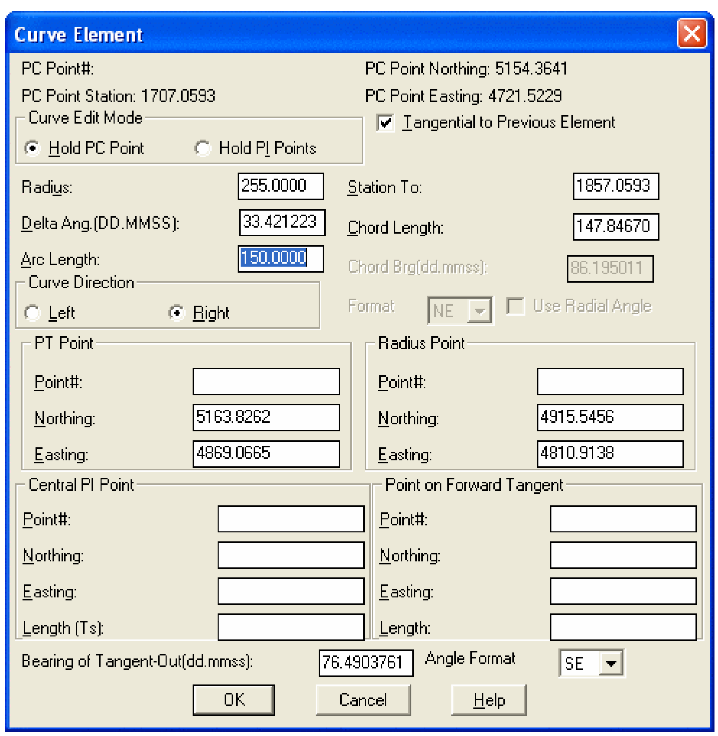


When OK is clicked, the routine will add the Line element as the first in the list of complete centerline elements. Next up is Curve-Spiral-Curve. Click Add.



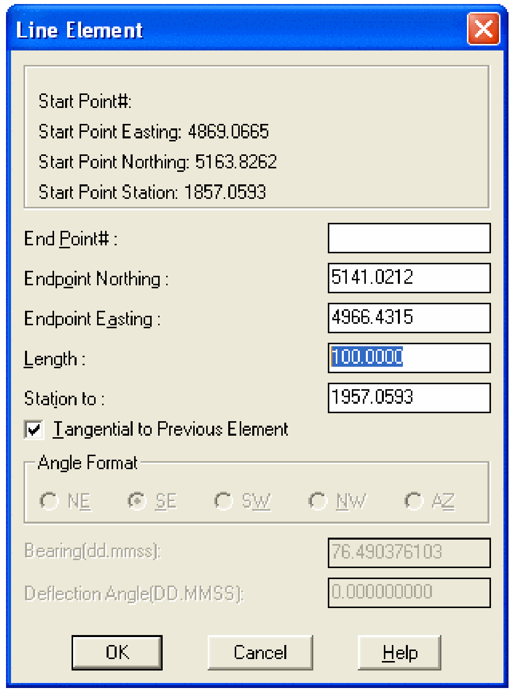
Spiral Segment: Though the dialog is complex (for total flexibility), the key on a typical symmetrical spiral curve is to enter four things: (1) the radius of the simple curve, (2) the spiral in and out lengths, (3) the tangent-out bearing and (4) the PI station (1835.67). Everything else will calculate when you press Enter for the PI station.

Curve Segment: Add the next element and select curve. The Curve dialog appears. The key is to enter the Radius Length (255), the Arc Length (150) and the Curve Direction. Everything else will calculate.

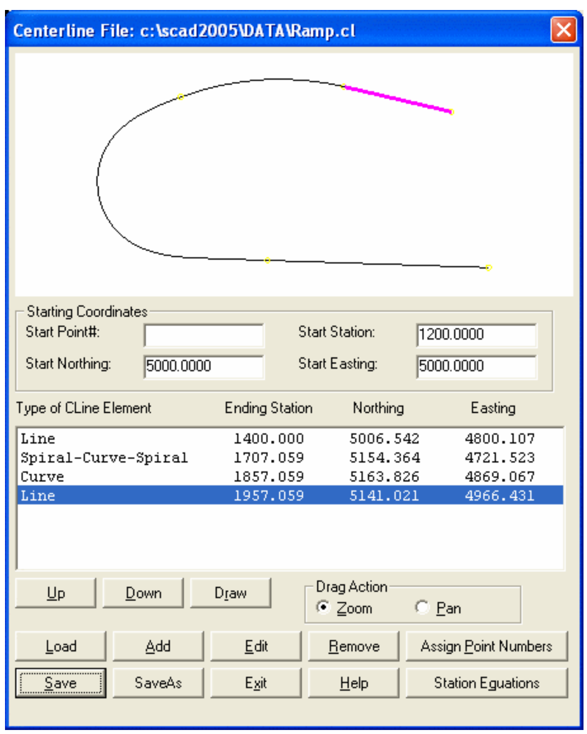


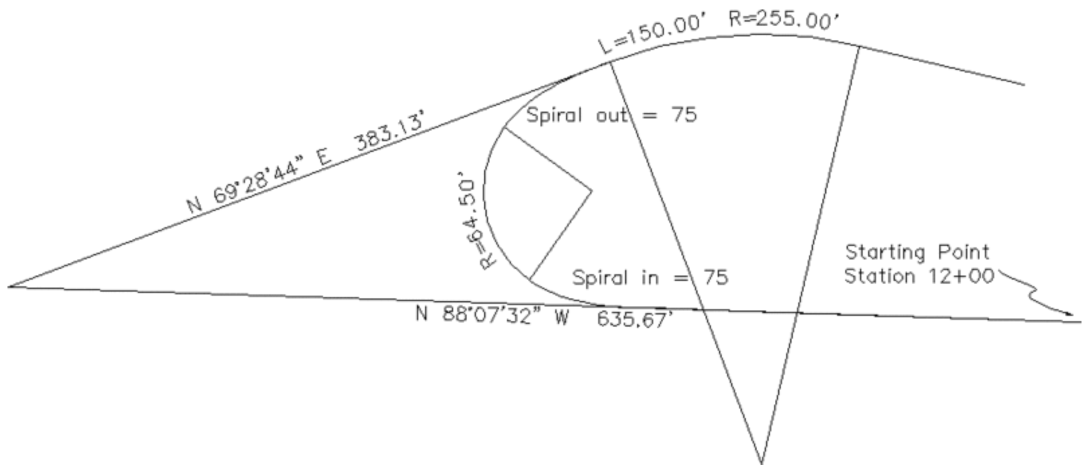
Final Line Segment: All you need to enter in the final dialog for the line (tangent) segment is its length. All other

items will calculate when you press Enter.



The completed centerline will appear as shown in the dialog and each element can be edited. Pick the Save button to store this centerline data to a .CL file.





Keyboard Command: cledit
Prerequisite: A CRD file to put points or take points from
File Name: \lsp\eworks.arx

Polyline to Centerline File

This command writes a centerline (.CL) file from a polyline. The northing and easting for each vertex of the polyline is written to the centerline file and each arc in the polyline becomes a circular curve.

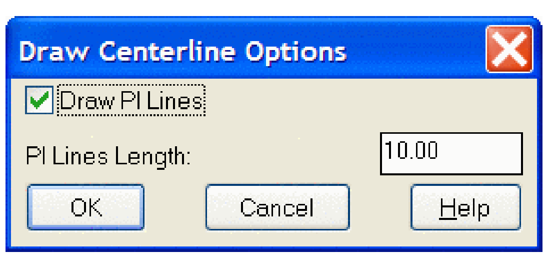
Prompts

Centerline file to Write dialog Enter the .CL file name to create.
Beginning station <0+00>: *press Enter* Or, type in the beginning station then press Enter.
Select polyline that represents centerline: *pick the polyline that represents your centerline*

Keyboard Command: clpline
Prerequisite: A polyline
File Name: \lsp\quikcl.lsp

Draw Centerline File

This command reads a centerline (.CL) file and plots it as a 2D polyline in the drawing at the proper coordinates. First you are prompted for the layer name for the polyline to be created. There is also an Options choice that allows you to specify whether to draw PI lines, and specify the length of same.



Next you are prompted for the file name of the centerline to plot.

The .CL file can be made with the following commands on the Design menu:*Polyline to CL File, Input-Edit Centerline* or *Design Centerline.* Drawing the centerline file is a way to check the .CL file data graphically for correctness. If a spiral exists in the .CL file, the spiral will be represented by polyline segments.

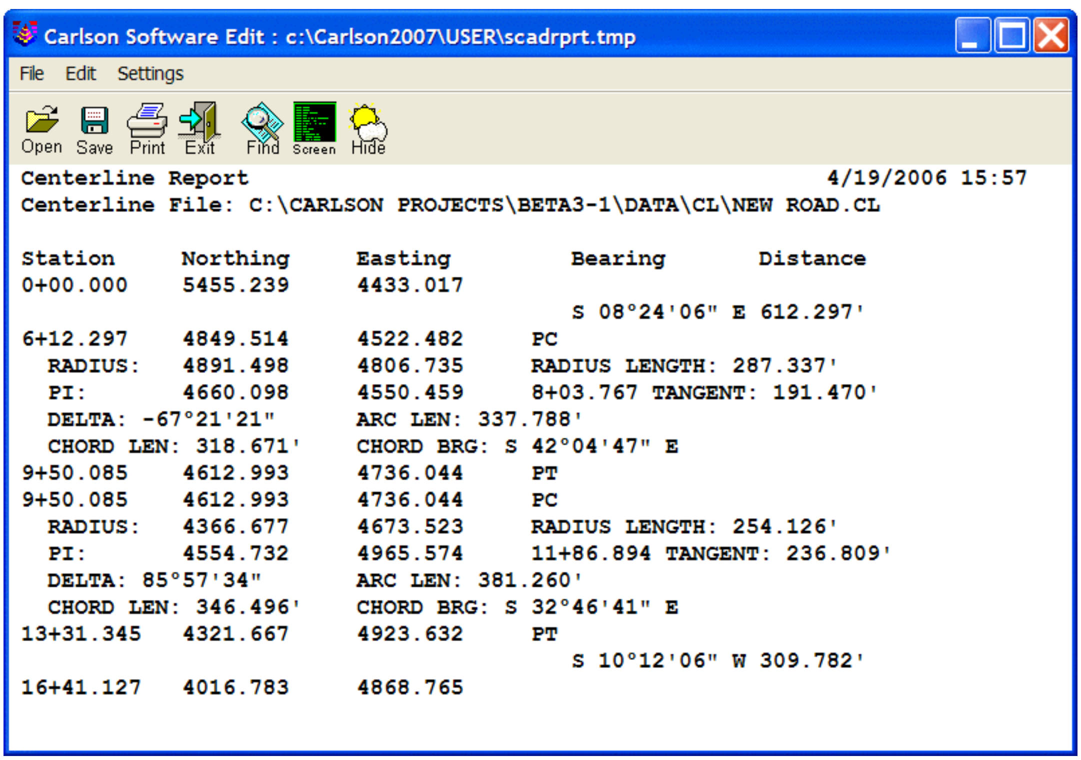
Prompts

Options/Layer Name for Centerline <CLINE>: *press Enter*Enter the layer name to plot the polyline on.
Centerline File to Plot file selection dialog Select the .CL file name to read and plot.

Keyboard Command: cl2pline
Prerequisite: a centerline file
File Name: \lsp\clpline.lsp

Centerline Report

This command reads a centerline file and creates a report in the standard report viewer which can be written to a file, a printer, or to your drawing. If the centerline file contains point numbers, then the report will include these point numbers. If station equations are found, they are noted at the top of the report. Here is an example report:




```
Centerline Report                                              4/19/2006 15:57
Centerline File: C:\CARLSON PROJECTS\BETA3-1\DATA\CL\NEW ROAD.CL

Station     Northing     Easting         Bearing       Distance
0+00.000    5455.239     4433.017
                                         S 08°24'06" E 612.297'
6+12.297    4849.514     4522.482      PC
  RADIUS:   4891.498     4806.735      RADIUS LENGTH: 287.337'
  PI:       4660.098     4550.459      8+03.767 TANGENT: 191.470'
  DELTA: -67°21'21"      ARC LEN: 337.788'
  CHORD LEN: 318.671'    CHORD BRG: S 42°04'47" E
9+50.085    4612.993     4736.044      PT
9+50.085    4612.993     4736.044      PC
  RADIUS:   4366.677     4673.523      RADIUS LENGTH: 254.126'
  PI:       4554.732     4965.574      11+86.894 TANGENT: 236.809'
  DELTA: 85°57'34"       ARC LEN: 381.260'
  CHORD LEN: 346.496'    CHORD BRG: S 32°46'41" E
13+31.345   4321.667     4923.632      PT
                                         S 10°12'06" W 309.782'
16+41.127   4016.783     4868.765
```

Keyboard Command: clreport
Prerequisite: A centerline (.CL) file
File Name: \lsp\eworks.arx

Import Centerline

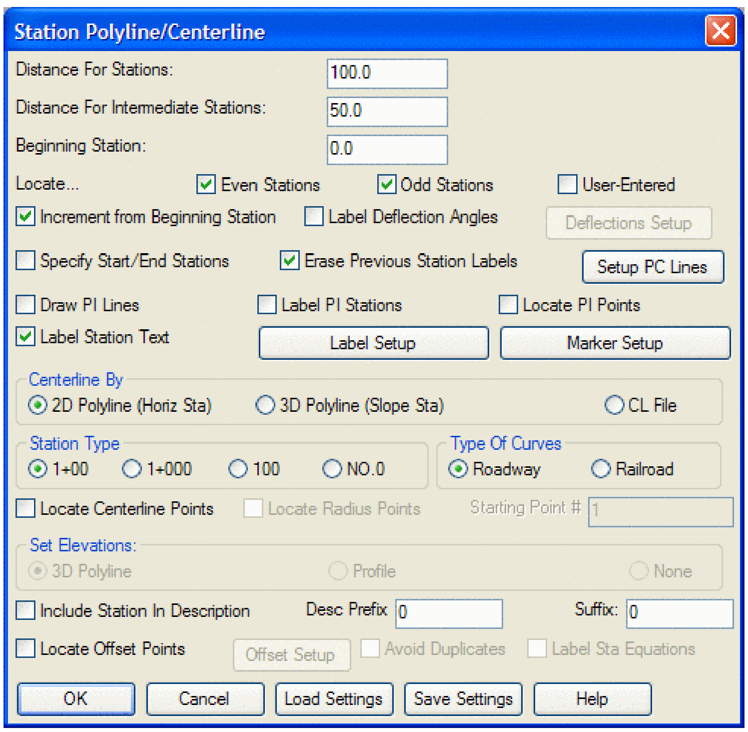
Function

This command converts Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, and Leica road files into Carlson TakeOff centerline (.CL) files.

Prerequisite: a Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, or Leica road file

Station Polyline/Centerline

This command will station a polyline or centerline file at a given interval distance. The options for this command are set in the dialog shown below. After setting the options, click OK on the dialog and then pick the polyline or select the centerline file. All settings can be saved as (.STA) files and loaded for reuse, and for storing multiple stationing schemes. Polyline/Centerline station labels are also dynamic, and so will update when changes are made in the geometry.



Distance for Stations is the primary interval for stationing.

Distance for Intermediate Stations is the intermediate interval for stationing.

Beginning Station is the beginning station of the centerline for stationing.

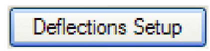
Locate Even Stations labels the stations at the distance interval (i.e. 2+00, 3+00, etc.).

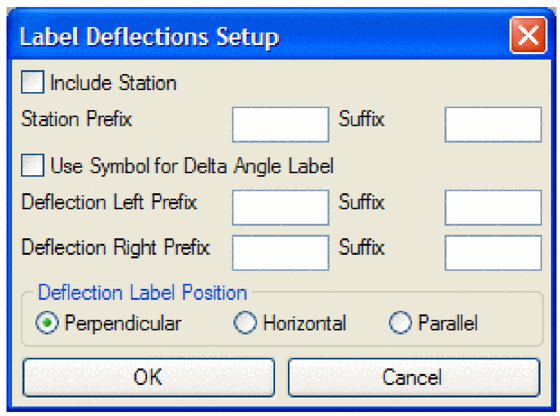
Locate Odd Stations labels the non-interval stations at the polyline/centerline end points and PC and PT points.

Locate User-Entered prompts you for individual stations to label.

Without the **Increment Station Labels from Beginning Station** option, the program increments the station labels from zero. For example, if the station interval is 100 and the polyline starting station is 145, then the program will label 2+00, 3+00, etc. With this option active, the station labels are incremented from the starting station. In this example, the program would then label 2+45, 3+45, etc.

Label Deflection Angles adds this annotation to the stationing. Settings for this are specified in the **Label Deflections Setup**, accessed by the **Deflections Setup** button.

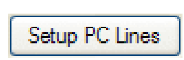


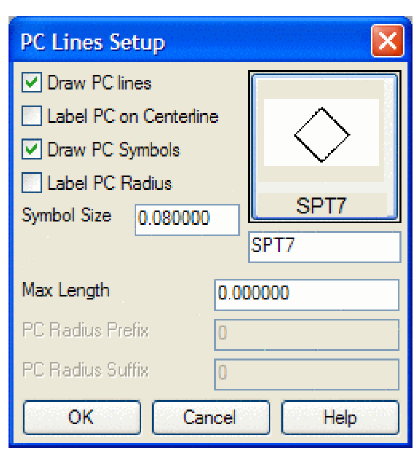


When **Specify Start/End Stations** is checked, only the stations between and including the specified starting and ending stations will be labeled. If locate centerline points and offset points are toggled on, only points within the specified stations will be located.

When **Erase Previous Station Labels** is checked, previous station labels are erased when new ones are generated.

The **Setup PC Lines** button accesses the **PC Lines Setup** dialog, where settings are controlled for lines and/or symbols and/or labels at the starting and ending (PC and PT) stations of an arc of the polyline/centerline.





Draw PC Lines controls whether lines are drawn from the PC and PT points.

When **Label PC On Centerline** is checked, the station of the PC and PT will be labeled on the centerline as well as the PC and PT lines. When not checked only the PC and PT lines will be labeled.

Draw PC Symbols controls whether symbols are placed at these locations. If checked, the desired symbol is selected by picking on the box to the right.

Label PC Radius controls whether this point is labeled.

Max Length controls the maximum length for the PC lines to be drawn described above.

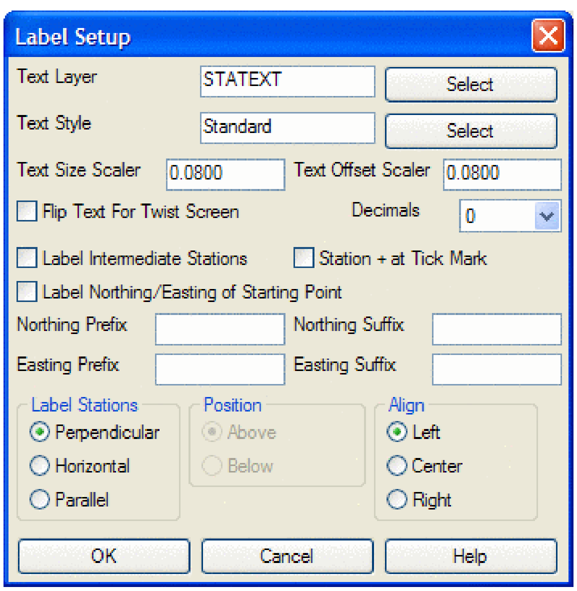
Back in the main Station Polyline/Centerline dialog box:

Draw PI Lines draws a 2 segment polyline in both tangent directions from the PI as a marker for the PI.

When **Label PI Stations** is checked, the PI station is labeled at the PI point.

When **Locate PI Points** is checked a point will be created at the PI of a horizontal curve graphically and written to the active coordinate file.

When **Label Station Text** is checked, this command places station text along the polyline at the angle of the corresponding segment. After toggling this option on, the *Label Setup* button will become available for selection. Select it to configure the label settings as desired. Select the *Marker Setup* options to modify the size of the markers for certain types of stations. See definitions following the dialog box.



Label Setup

- **Text Layer** is the user-specified layer for text labels to be drawn on.

- **Text Style** is the user-specified text style for labels.

- **Decimals** determines the number of decimal places of the stationing labels to be drawn for the odd stations and user entered stations only.

- **Text Size Scaler** determines the size of the station labels. This value multiplied by the horizontal scale setting in Drawing Setup results in the size of the label. For example, if the horizontal scale is set to 100 and the text size scaler is set to 0.10, the station labels will be 10 units.
- **Text Offset Scaler** works like text size scaler above controlling the distance the text labels will be offset from the centerline.

- If the **Flip Text For Twist Screen** setting is checked and the drawing has been twisted using the twist screen command, the label text will be flipped to read in the proper direction of the stationing.

- **Label Intermediate Stations:** If the intermediate distance is the same as the station distance then no intermediate station ticks or labels will be drawn. For example, with the above entries and 0+00 for the first station the stations will be labeled with descriptions as follows: 0+00 0+50 1+00 1+50, etc.

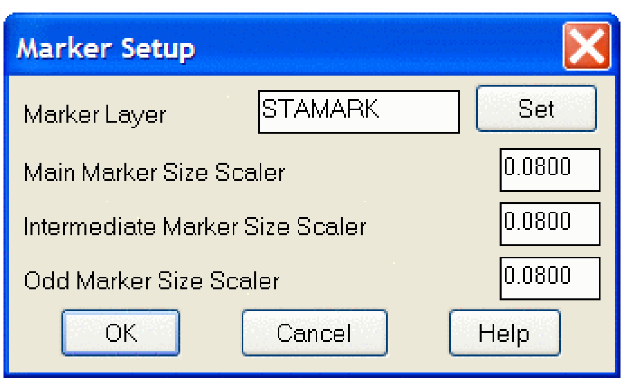
- **Station + at Tick Mark** labels the station text along the polyline with the '+' of the station text at the station's location on the polyline. See Marker Set up for marker size manipulation settings.
- **Label Northing/Easting of Starting Point** adds this label information, including prefixes and/or suffixes as specified.

- Use **Label Stations** to specify whether to label the stations perpendicular or parallel to the centerline.

- Specify the **Position** of the station labels, either above or below the centerline. This is only available when labeling stations using the parallel option.

- **Align** determines the alignment of the station label, either left or centerline, centered along the centerline or to the right of the centerline. This option is only available when using the perpendicular option for station labels.

The **Marker Setup** options control the size of markers for different station types as well as the layer the markers will be drawn on.



Specify whether to define the **Centerline By** picking a 2D polyline or 3D polyline in the drawing or selecting a centerline (.CL) file.

- Using a **2D Polyline** will result in horizontal distance stationing along the polyline.
- Using a **3D Polyline** will result in the slope distance stationing along the polyline.
- Using a **CL File** will result in horizontal distance stations as with the 2D Polyline option only a prompt for the centerline to use will display.

Use **Station Type** to specify the stationing format to use.

Use **Type of Curves** to specify whether you are labeling a roadway curve (arc definition) or railroad curve (chord definition).

Locate Centerline Points will locate points and store them in the current CooRDinate file.

Locate Radius Points will locate the radius points of any arc segments.

Starting Point Number determines the starting point number for the points to be located.

There are two ways to **Set Elevations** for the centerline points and offset points to be created.

- The **3D Polyline** option gets the elevation of the point from a specified 3D Polyline within the drawing.
- The **Profile** option will determine the elevation of the point based upon the same station in the profile file. You will be prompted for the profile file to read for the elevation reference.
- With the **None** option selected, no elevations will be determined for the points.

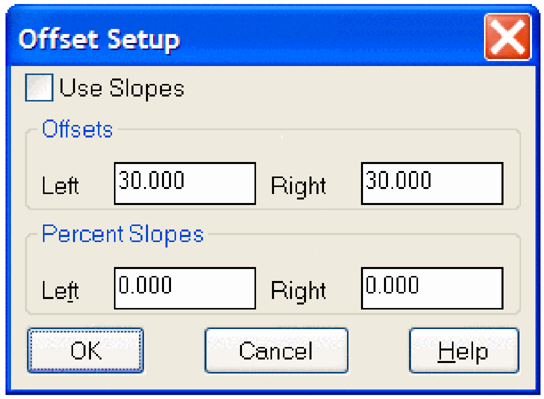
When **Include Station in Description** is checked, the station along the centerline will be included in the resulting offset point.

Description Prefix is an optional user-specified prefix to be added to the point description.

Description Suffix is an optional user-specified suffix to be added to the point description.

When **Label Sta Equations** is checked on any station equation, contained in a centerline (∗.cl) file will be labeled. This option is only available when stationing a centerline file (∗.cl).

Locate Offset Points will create points at the specified left and right offset distances from the centerline. Options for setting the elevations and descriptions of the points are available from the Offset Setup dialog.

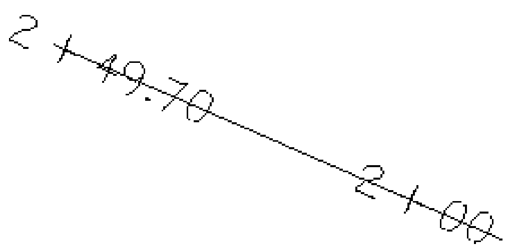


- When **Use Slopes** is on, it makes available the Percent Slopes fields for defining the slope from centerline both right and left for determining the elevations of the offset points.

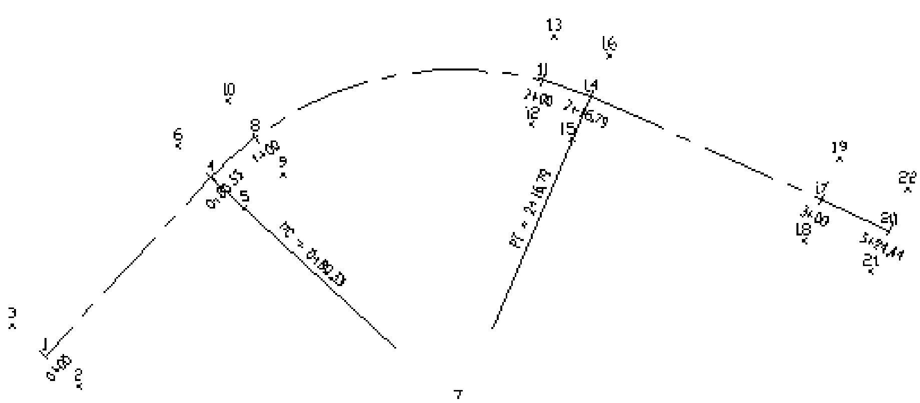
- Enter the desired **Offsets** left and right.
- Enter the desired **Percent Slopes** from centerline to the left and right offset points.

Prompts

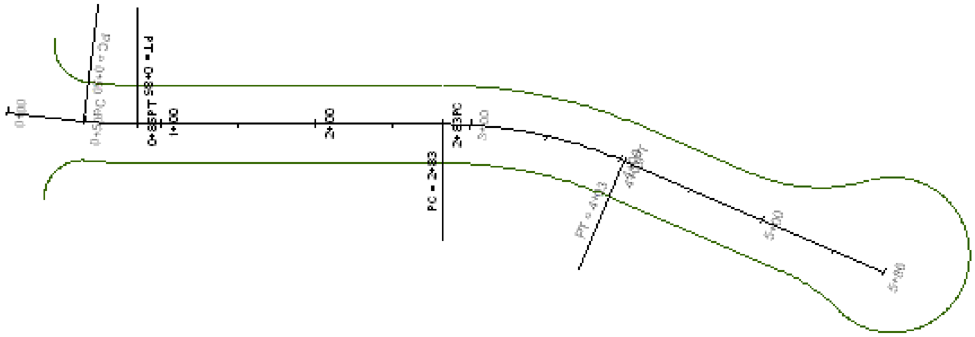
Station Polyline Dialog
Polyline should have been drawn in direction of increasing stations.
Select polyline that represents centerline: *select a polyline*



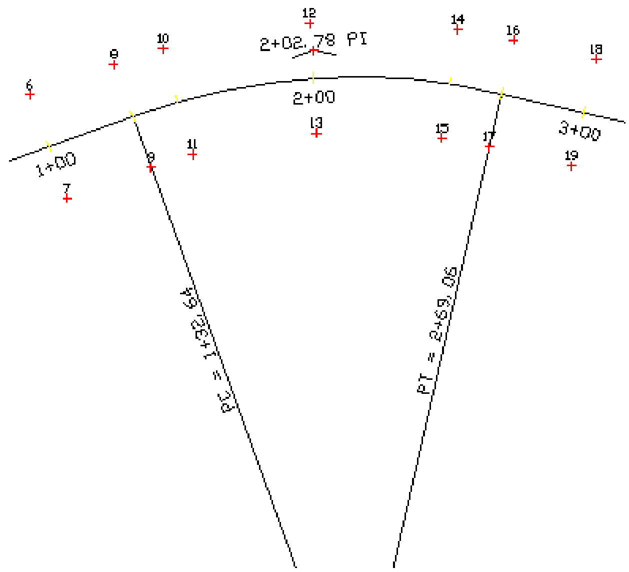
Closeup of Station + at Tick Mark option



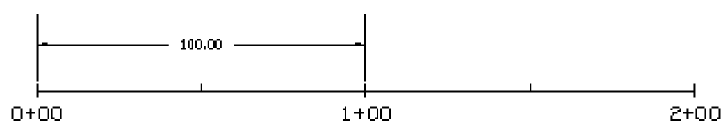
Labels with Label PC on Centerline checked on



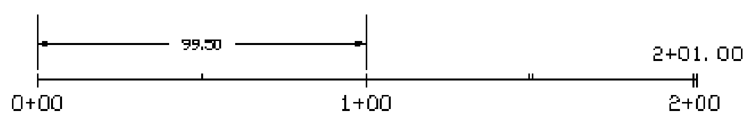
Labels set to perpendicular and Max Length of PC lines set to 75.0



Labels with Draw PI Lines, Label PI Stations and Locate PI Points all checked on



Labels using Centerline By 2D Polyline (Horizontal Station)

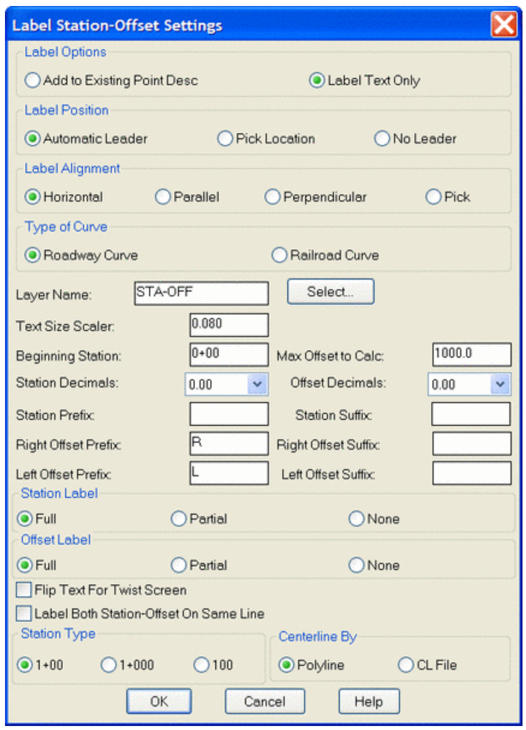


Labels using Centerline By 3D Polyline (Slope Station)

Keyboard Command: stapl
Prerequisite: A polyline or CL file
File Name: \lsp\staent.lsp

Label Station-Offset

This command will label the station and offset of a point relative to a centerline. A polyline that represents the centerline or a centerline (.CL) file is required before using this command. The points to label can either be picked on screen or specified by point number. As the crosshairs are moved, the station and offset of the current position are displayed in real-time in a small window (see example). This command starts with the Label Station-Offset Settings dialog.



Label Options: Specify whether to label the text only or add the label to the existing point description.

Label Position: Specify if the program should automatically position the label with a leader, let the user pick the location, or use no leader.

Label Alignment: Specify whether the labels should be Horizontal on the screen, Parallel to the Centerline, Perpendicular to the Centerline, or user-specified by Picking.

Type of Curve: Specify whether the centerline is for a roadway or railroad.

Layer Name: Specify the layer name for the labels.

Text Size Scaler: Determines the size of the labels. This value multiplied by the horizontal scale setting in *Drawing Setup* results in the size of the label. For example, if the horizontal scale is set to 100 and the text size scaler is set

to 0.10, the labels will be 10 units.

Text Offset Scaler: Determines the text offset. This value works the same way as the Text Size Scaler.

Beginning Station: Specify the beginning station of the centerline. The polyline should be drawn in the order of increasing stations. Not available when you use a centerline (.CL) file to define the centerline.

Max Offset to Calc: Specify the maximum offset to calculate.

Station Decimals: Specify the display precision for the station text.

Offset Decimals: Specify the display precision for the offset text.

Station Prefix: Specify an optional prefix for the station text.

Station Suffix: Specify an optional suffix for the station text.

Offset Prefix/Suffix: Specify an optional prefix and/or suffix for each offset.

Station Label: Choose between Full label (1+35.42), Partial label (+35.42) or no station label.

Offset Label: Choose between Fill label (L15.35), partial label (15.35) or no offset label.

Flip Text for Twist Screen: With this option checked, the text will be flipped as necessary to adjust for the use of Twist Screen.

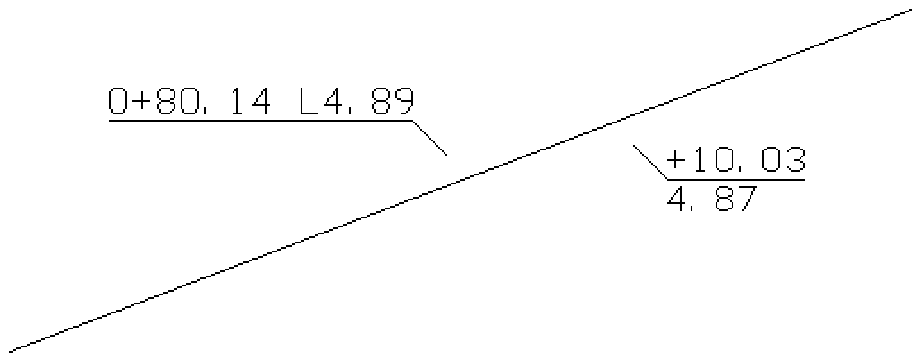
Label Both Station-Offset On Same Line: With this option checked, the station and offset label will be drawn on the same line.

Station Type: Specify the stationing format to use.

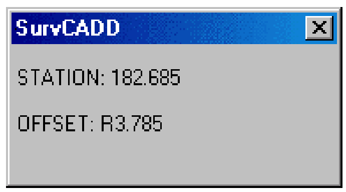
Centerline By: Specify whether to define the centerline by picking a polyline in the drawing or selecting a centerline (.CL) file.

Prompts

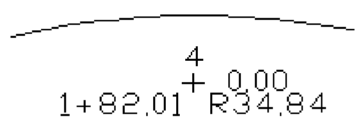
Label Station-Offset dialog
Polyline should have been drawn in direction of increasing stations.
Select Polyline Centerline: *pick the polyline centerline*
Pick point or point number (SS for Selection Set, G for Group, Enter to End): *pick a point*
Station on Line> + 2+10.91 Offset> 57.36 Right
Select point number to add station description to: *pick point number* This prompt will not appear if the L option, label only was selected.
Pick point or point number (Enter to End): *press Enter*



Top/Left Example: Label Text Only showing Full Labels and the Station/Offset on the same line
Bottom/Right Example: Partial Label



Real time display of station offset as you move the cursor

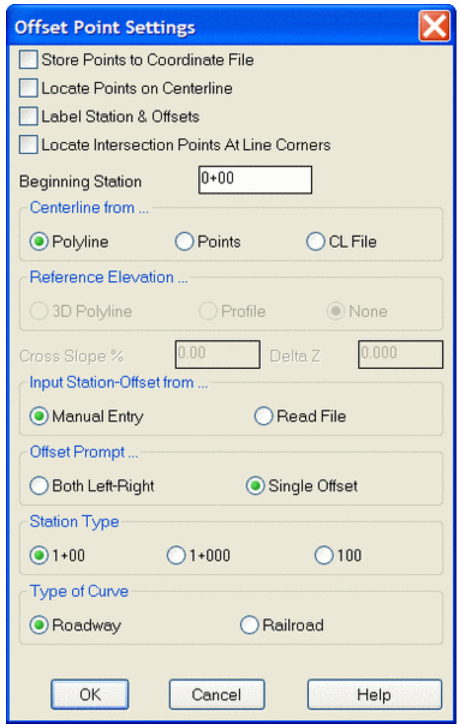


Add to Point Description format

Keyboard Command: offsta
Prerequisite: A polyline centerline
File Names: \lsp\inqoff.lsp, \lsp\scadcogo.dcl

Offset Point Entry

This command creates points along a centerline at specified stations and left and right offsets. The centerline can be defined by a polyline, centerline (.CL) file or two points.



The **Store Points to Coordinate File** option will store any points the current coordinate (.CRD) file. This includes centerline points and offset points.

When **Locate Points on Centerline** is checked, the program will locate points along the centerline, otherwise just the offset points will be created.

When **Label Stations & Offsets** is checked, the program will label the station-offset as the point description attribute.

When **Locate Intersection Points At Line Corners** is checked, the program will locate points along the centerline at the intersection points of selected lines with that of the centerline. This routine is to be used along with Locate Points on Centerline. This is a good option to use when the exact station of where the offset points are to be created is not known but is referenced by an existing line on the drawing.

Beginning Station: Enter the Beginning Station of the Centerline.

Use **Centerline from** to specify whether to define the centerline by picking a polyline in the drawing, selecting a centerline (.CL) file, or using 2 points.

Use **Reference Elevation** to assign elevations to the points created when locating points on the centerline of offset points. When using a 3D Polyline for the elevation reference, points will be created at the station entered and the offsets specified with the elevation of the same station along the 3D polyline. The Profile option will do the same as the 3D Polyline option only it will use a profile file for the elevation reference. You will be prompted for the profile to use for the elevation reference. None simply creates 2d point data on elevation zero. The Reference Elevation option is good for creating points along the centerline for final grade elevation points. *Profile to 3D polyline* can be used to transfer the profile data to the polyline before calculating the final grade points.

Cross Slope %: This option is used to alter the elevations of the new points by applying either a Cross Slope calculation or a Delta Z variable.

The Manual Entry option in **Input Station-Offset from** will prompt for the station and offset distances. The Read File option will read the stations and offsets from a text file. The text file format is comma delimited with point number, station, offset and elevation. The station should be just the station number without the '+' (i.e. 250 instead of 2+50). The elevation is optional. The Read File option is a quick routine to convert a station-offset data file into coordinates.

When **Offset Prompt** is set to Both Left-Right, the program will prompt for left and right offsets. If you respond to an offset prompt with zero (0), no offset point is created. The Single Offset option will prompt for one offset per station. Enter a right offset with a positive value and a left offset as a negative value.

Use **Station Type** to specify the stationing format to use.

Use **Type of Curve** to specify whether the curves are for a roadway or railroad.

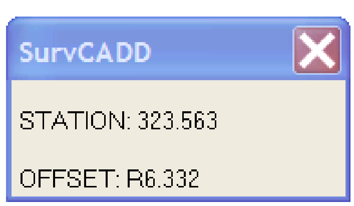
Prompts

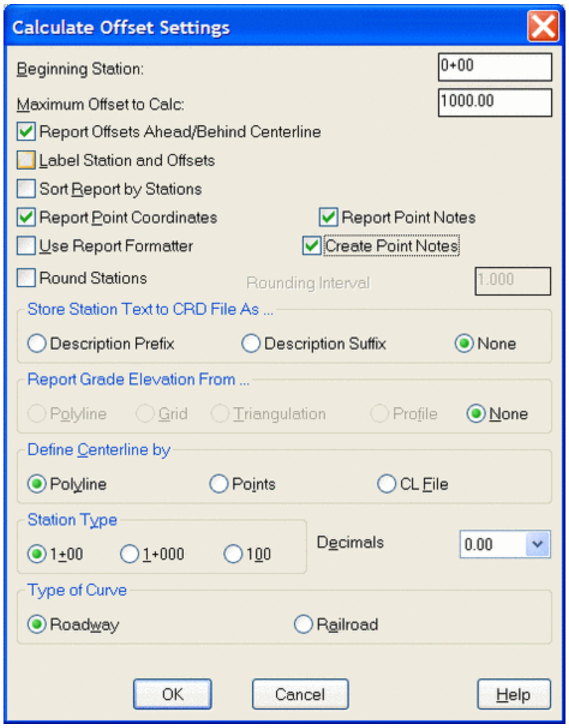
Offset Point Settings Dialog
Polyline should have been drawn in direction of increasing stations.
Select Polyline near endpoint which defines first station.
[nea on] Select Polyline to Station-Measure: *select a polyline*
(5309.0 4845.0) Station: 0.00
(5526.0 4917.0) Station: 228.63
Distance from beginning station along centerline (Enter to end): *110*
Starting Segment Station: 0.0 Ending Segment Station: 228.633
Working Line segment...(5413.4 4879.64 0.0)
Left offset distance <10.0>: *15*
Right offset distance <15.0>: *20*
Distance from beginning station along centerline (Enter to end): *press Enter*

Keyboard Command: offpts
Prerequisite: A centerline (.CL) file, polyline, or two points
File Name: \lsp\offpnts.lsp

Calculate Offsets

This command calculates the station and offsets of point coordinates relative to a centerline. The points to calculate can be stored in a coordinate (.CRD) file or picked on the screen. As the crosshairs are moved, the station and offset of the current position are displayed in real-time in a small window (see example).





Beginning Station: Specify the beginning station of the centerline. The polyline should be drawn in the order of increasing stations. Not available when you use a centerline (.CL) file to define the centerline.

Maximum Offset to Calc: This is the maximum distance from the Centerline for which offsets are calculated.

Report Offsets Ahead/Behind Centerline: When checked, this option shows offsets for points or picked points located before the beginning station and after the ending station of the centerline.

Label Station and Offsets: When checked, the station offsets will be labeled in the drawing.

Sort Report by Stations: When checked, this option will report the station-offsets in station order no matter what order the points were calculated.

Report Point Coordinates: When checked, this option will include the point northing and easting in the report.

Report Point Notes: When checked point notes will be included on the calculate offset report.

Create Point Notes: When checked, the station and offset of the offset point will be created as notes and written to a note file (∗.not). This note file will have the same name as the crd file.

Use Report Formatter: When checked, the output of this command is directed to the Report Formatter which allows you to customize the layout of the report fields and can be used to output the data to Microsoft® Excel or Microsoft® Access. You must check this option on in order to use the Report Grade Elevation From option.

Round Stations: When checked, this option will round the stations for the selected points on the report to the Rounding Interval specified. For example if an offset point is located at station 1+01, and the rounding interval is set to 10, then the report will show the offset point at station 1+00.

Store Station Text to CRD File: When checked, the station offset text is appended to point numbers that are selected.

Report Grade Elevation From: When checked, this option will calculate an elevation for each point from a 3D polyline, grid file (.grd) or triangulation (.flt) file. To Use this option, the *Report Formatter* must be toggled on. The grade elevation is reported and compared with the point elevation to report the cut/fill. For the 3D polyline option, the grade elevation is calculated by finding the elevation at the point on the 3D polyline that is the nearest perpendicular position from the offset point. The 3D polyline that is used for elevations does not need to be the same polyline that is used as the centerline for the station-offset calculations.

Define Centerline by: Specify whether to define the centerline by picking a polyline in the drawing, selecting a centerline (.CL) file, or using 2 points.

Station Type: Specify the stationing format to use.

Decimals: Specify the display precision for the stations and offsets.

Type of Curve: Specify whether the curves are for a roadway or railroad.

Prompts

Calculate Offset Settings Dialog
Polyline should have been drawn in direction of increasing stations.
Select Polyline near endpoint which defines first station.
[nea on] Select Polyline Centerline: *select polyline centerline*
(5309.0 4845.0) Station: 0.00
(5526.0 4917.0) Station: 228.63
PtNo. North(y) East(x) Elev(z) Description
140 4889.13 5410.25 0.00 1+10.00L10.00
Station on Line> 1+10.00 Offset> 10.00 Left
PtNo. North(y) East(x) Elev(z) Description
141 4870.15 5416.55 0.00 1+10.00R10.00
Station on Line> 1+10.00 Offset> 10.00 Right
+ before station denotes point is ahead of line segment, - denotes beyond.
Pick point or point numbers (Enter to End): *22-28*

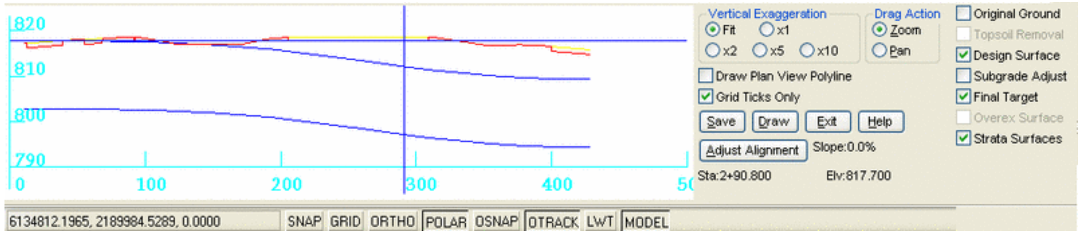
```
  Station  Offset  Description  Elev  Pt#  North    East
4+95.89L  15.48   Catch Basin  0.00  22   4811.00  4454.00
5+78.43L  58.18   Power Pole   0.00  23   4839.00  4548.00
6+77.26L  57.28   Power Pole   0.00  24   4868.00  4656.00
9+01.55R  16.81   Catch Basin  0.00  25   4745.00  4887.00
10+50.51L 25.39   Traffic Sign 0.00  27   4872.00  5043.00
4+03.48R 22.15    Light Pole   0.00  28   4657.00  4454.00
```

Pick point or point numbers (Enter to End): *press Enter*

Keyboard Command: calcoff
Prerequisite: A centerline (.CL) file, polyline or two points
File Name: \lsp\pstaoff.lsp

Quick Profile From Surface Entities

This command allows you to create, view, edit, and report profiles from the TakeOff surfaces.



Pick starting point (CL-Centerline,P-Polyline): To make a profile you need to define the alignment by: 1) picking points on the screen; 2) typing in CL in the command prompt, and selecting a centerline file; or 3) typing in P and choosing a polyline from the screen. After doing so, the above profile viewer is created.

The far right dialog box allows you to toggle on and off different Surfaces to view in the profile viewer including: Original Ground, Topsoil Removal, Topsoil Replacement, Design Surface, Final Subgrade, Overex Surface, Strata Surfaces. If a surface is not defined in the current TakeOff project, like Topsoil Removal in this example, than you will not have the option to display it. In this example, the three Surfaces that can be displayed, Original Ground, Design Surface, and Final Subgrade, are displayed in the profile viewer.

When you move the cursor around the profile viewer a crosshair follows along the surface and reports the Station, Slope %, and Elevation at each point. It is displayed towards the bottom-right side of the screen next to Adjust Alignment. In this example the station is 2+16.650, the Slope is -5.6%, and the Elevation is 818.133. A crosshair can been seen in the profile drawing and along the alignment in the main drawing as well.

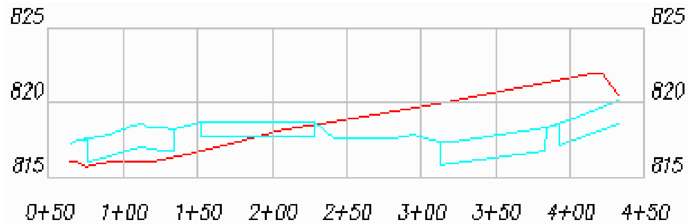
Vertical Exaggeration: x1 is the actual appearance of the surface(s). Depending on the flatness of the surface(s), you can select x2, x5, x10 vertical exaggerations to better see the elevation differentiation and different surfaces. The option Fit automatically exaggerates the vertical to best fit the profile viewer.

Drag Action: This dialog allows you to zoom in and out, and pan around the profile. To zoom in click and drag up, to zoom out click and drag down. To Pan, click amd drag the direction you want to move.

The Adjust Alignment icon allows to pick the polyline or centerline that you used and move it to your liking. If you selected an endpoint vertex, you can pivot that vertex around 360 degrees and the profiles will update in real time. This is helpful when checking for spikes. If you select the middle vertex then you can shift the entire centerline around.

If you created a profile alignment by picking points and you want to save that polyline you created then toggle on Draw Plan View Polyline. If you do not choose Draw Plan View Polyline than the polyline will be lost when you exit out of the Quick Profile command. Grid Ticks Only marks elevations and distances but does not draw them into grids.

The Save icon allows you to save the profile as a (.pro) file by whatever name you give it. The Draw icon allows you to draw the profile right on your drawing. Set the layer name, vertical and horizontal scale as desired, pick a starting point to draw, and the profile is created. Note: the below example has a veritcal scale of 5 feet per grid and a horizontal scale of 50 feet per grid.



Prompts

Command:
QUICKPRO
Pick starting point (CL-Centerline,P-Polyline): p

Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>:
Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Prerequisite: a surface

Keyboard Command: QUICKPRO

Profile From Existing Surface

Function

This command will create a profile file (.pro) for the existing surface. To define the profile alignment, type in CL in the command prompt, and select a centerline file, or pick the polyline from the screen. This will create the profile file. You can now use the other Profile commands to draw, edit and report from this profile.

Prompts

Command: progrid2

Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>:
Enter the starting station <0.0>:

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Found 19 profile points.

Prerequisite: a surface

Keyboard Command: progrid2

Profile From Design Surface

Function

This command will create a profile file (.pro) for the design surface. To define the profile alignment, type in CL in the command prompt, and select a centerline file, or pick the polyline you want to use from the screen. This will create the profile file. You can now use the other Profile commands to draw, edit, and report from this profile.

Prompts

Command: progrid3

Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>:
Enter the starting station <0.0>:

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Found 19 profile points.

Prerequisite: a surface

Keyboard Command: progrid3

Design Road Profile

Function

This command is for simultaneously creating a .pro file and drawing the road profile. The procedure is to first specify the on-screen grid and then enter or pick the stations and elevations.

Once two segments have been entered, you will be prompted for the vertical length. If you don't want a vertical curve, enter 0. Otherwise you can directly enter the vertical curve, or enter the sight distance or the K-value from which the vertical curve is calculated. The vertical curve can also be specified to pass through a point or do a best fit through multiple points. This through point option would be useful for hitting an existing feature such as a driveway on the vertical curve. Unequal vertical curves is another option where the vertical curve length going into the PVI

differs from the length leaving the PVI. Before using your entry, the vertical curve, sight distance, and K-value are displayed. Object height and eye height are two variables that effect the vertical curve.

Notice that the station, elevation, and slope at the current position of your cursor crosshairs are displayed in real-time in a small dialog.

Prompts

File Selection dialog Specify a profile file to create.

Profile Settings dialog

Station of first PVI or pick a point: *0*

Elevation of PVI: *565*

Station of second PVI or pick a point ('U' to Undo): *200*

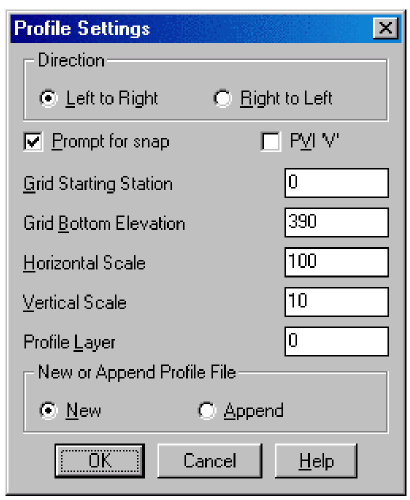
Percent grade entry/Ratio/<Elevation of PVI>: *575*

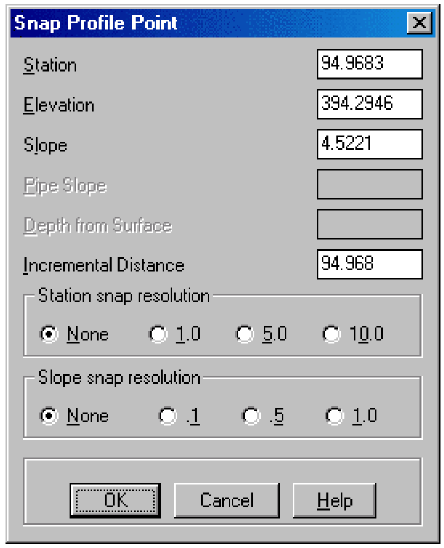
Station of next PVI or pick a point ('U' to Undo, Enter to End): *pick a point*

Snap PVI dialog

The Snap PVI dialog box appears when you pick a point (if the Prompt for Snap option in the Profile Settings dialog is selected). The station and slope may be changed to the nearest snap value. The elevation is the free variable and it will change to compensate for any snap. To change the elevation, select the elevation edit box and enter the new value.

View Table/Unequal/Through pt/Sight Distance/K-value/<Length of Vertical Curve>: *100*





For Crest with Sight Distance>VC and Vertical Curve => 100.00

Sight Distance => 87.30, K-value => 44.1

Use these values (<Y>/N)? *Press Enter*

Station of next PVI or pick a point ('U' to Undo, Enter to End): *press Enter*

Vertical Curve Text Options dialog box

Pick vertical position for VC text: *Pick a position above the profile grid.*

Prerequisite: A profile grid

Keyboard Command: road

Design Sewer Profile

Function

This command creates a sewer profile (.PRO) file and draws it on the screen. It requires that a grid is already drawn. It begins with the Profile Sewer Settings dialog box.

Prompts

Bottom Manhole Width: Specify the size for the bottom of manholes. Not available when Profile Type is set to pipe.

Max Pipe Length: Specify the maximum limit for the distance between manholes.

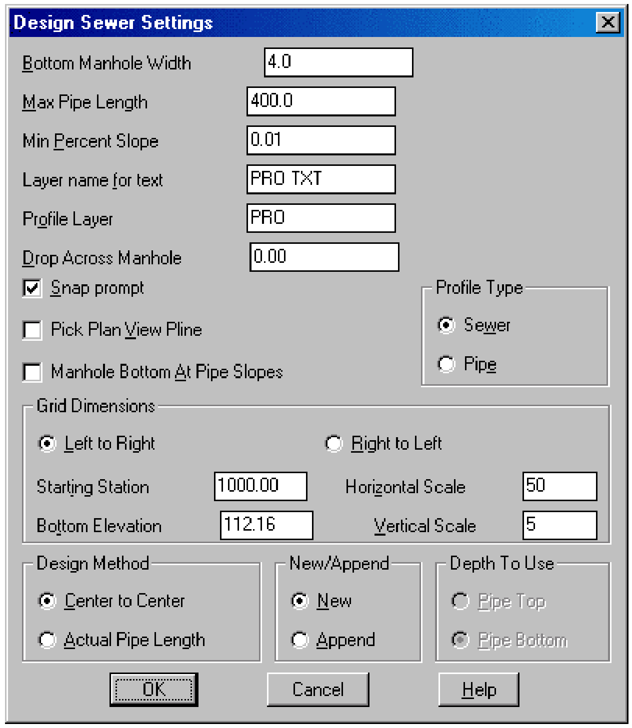
Min Percent Slope: Specify the minimum slope (absolute value) between manholes.

Layer name for text: Specify the layer name for annotation. If you enter a layer that does not exist, it will be created.

Profile Layer: Specify the layer name for pipes and manholes. If you enter a layer that does not exist, it will be created.

Drop Across Manhole: Specify the amount the elevation drop across the manhole in the direction of the profile. Will accept a negative a value. Not available when Profile Type is set to pipe.

Snap Prompt: Activates the PVI Snap dialog box. See below for description.



Pick Plan View Polyline: Allows you to select a polyline from plan view that represents the sewer center-line.

Manhole Bottom At Pipe Slopes: When checked, the manhole bottom will be drawn level with the pipe slope.

Profile Type: Choose between Sewer profile or Pipe profile. Pipe profile do not include manholes.

Grid Dimensions: Specify the grid dimensions on which the sewer will be designed.

Design Method: Choose whether distances specified are center or manhole to center of manhole or actual pipe length. Not available when Profile Type is set to pipe.

New/Append: Choose between creating a new profile (.PRO) file or appending an existing file.

Depth to Use: Choose between specifying pipe top or pipe bottom elevations. Not available when Profile Type is set to sewer.

File Selection dialog

Choose a new profile file name to create.

Pick Lower Left Grid Corner <5000.0,5000.0>[endp on]: *Pick the corner*

Select existing ground polyline or ENTER for none: *You may optionally pick a polyline to use for calculating the depth from the surface as the sewer stations are entered.*

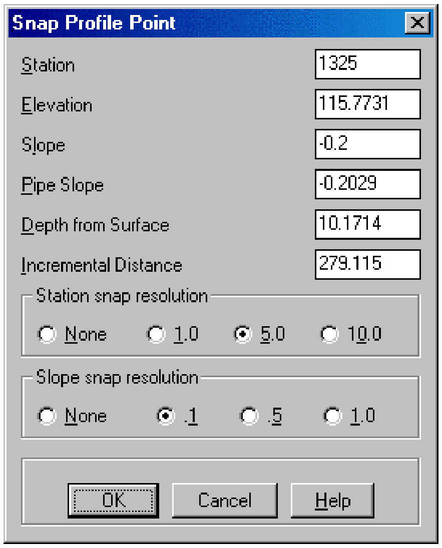
Station of first manhole or pick point: *0*

Invert Elevation of Manhole: *910*

Enter the step up/down in feet <0.00>: *Press Enter*

Station of second manhole or pick a point (U,E,D,Help): *pick a point*

If the Prompt for Snap option was selected in the main dialog, then the Snap Profile Point dialog appears here. The station and slope may be changed to the nearest snap value. The elevation is the free variable and it will change to compensate for any snap. To change the elevation, select the elevation edit box and enter the new value.

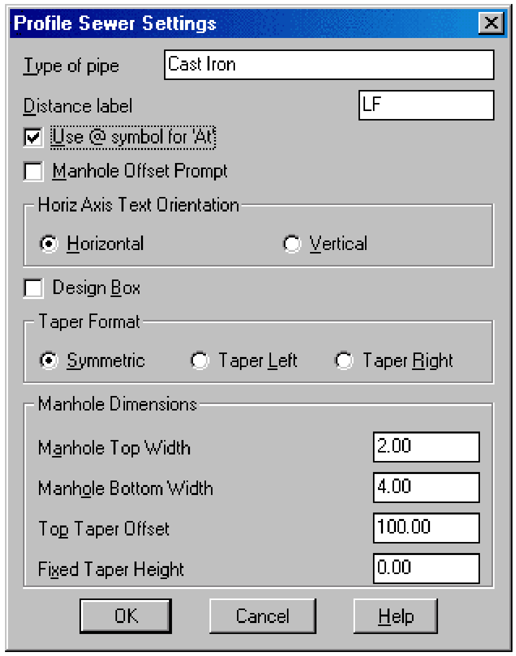


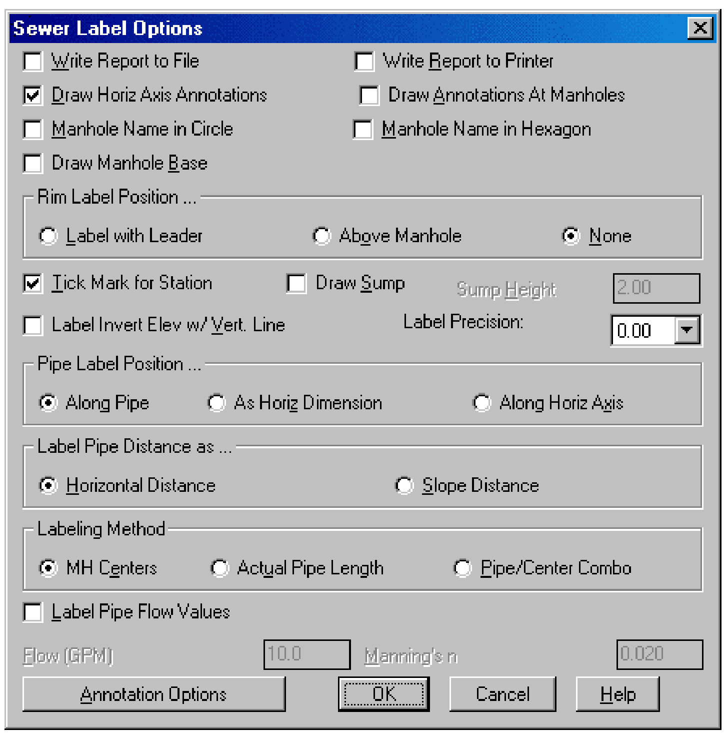
Enter the step up/down in feet <0.00>: *Press Enter*

Size of pipe in inches <10.0>: *8.0*
Station of next manhole or pick a point (U,E,D,Help): *Press Enter*

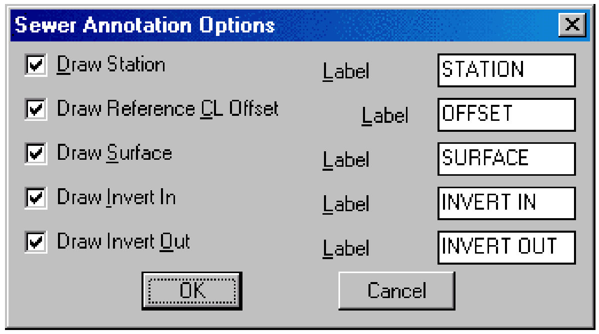
Profile Sewer Settings dialog

Sewer Label Options dialog





Sewer Annotation Options dialog (displayed by pressing the Annotation Options button)



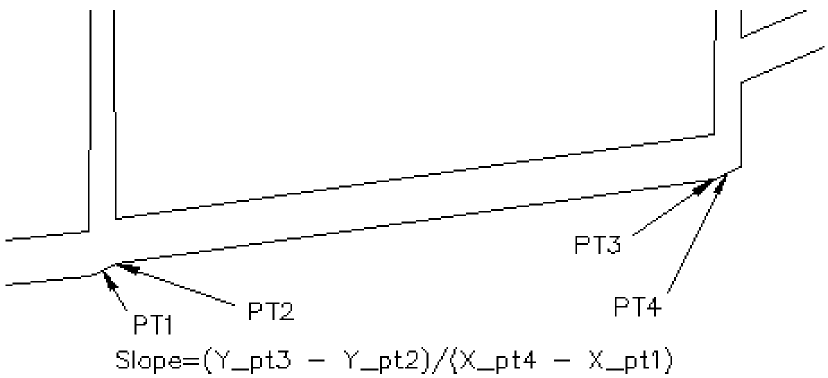
Select existing ground polyline: *Pick a polyline or press Enter to be prompted for each manhole surface elevation.* This prompt only appears if no ground polyline was selected above.

Manhole No. 1 label [MH #1]: *Press Enter*

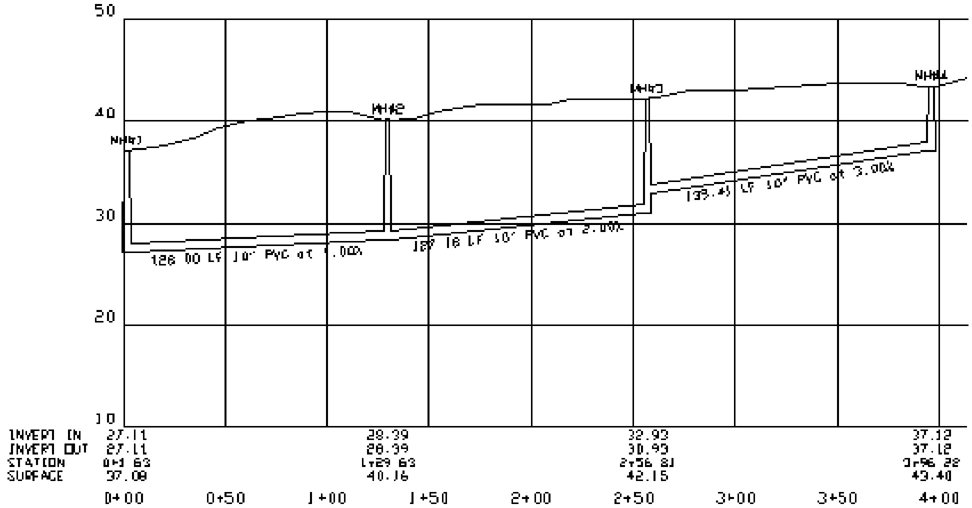
Manhole No. 2 label [MH #2]: *Press Enter*

Prerequisite: A profile grid

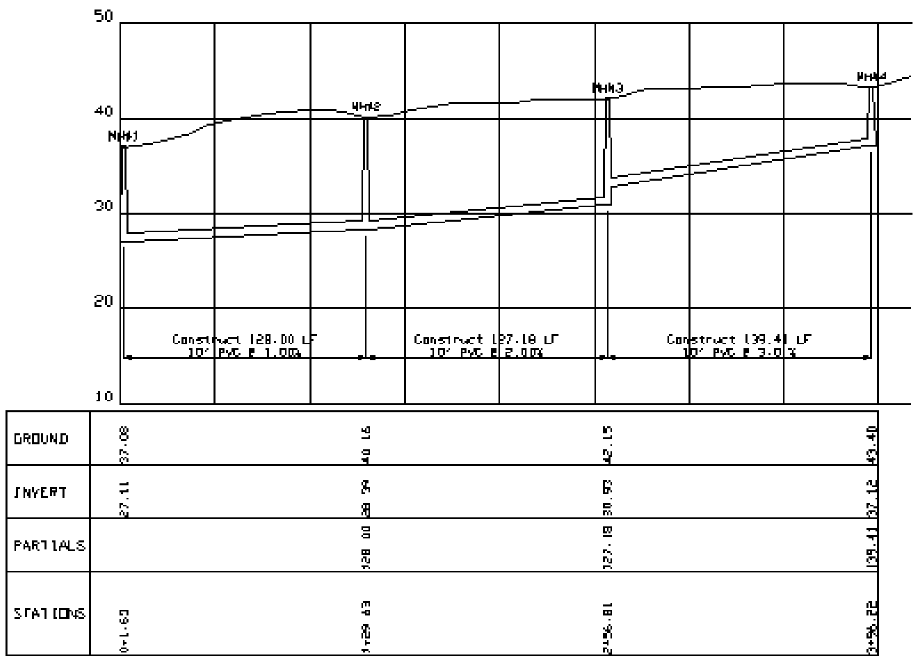
Keyboard Command: sewer



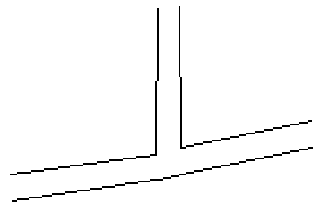
Pipe/Center Combo Labeling Method calculates the slope as the elevation difference from the edge of the pipe divided by the distance between the manhole centers.

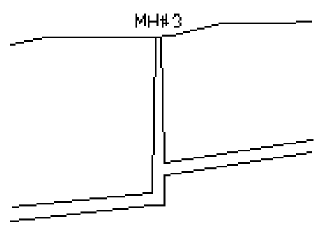


Example of sewer profile and surface profile

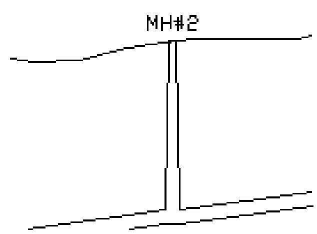


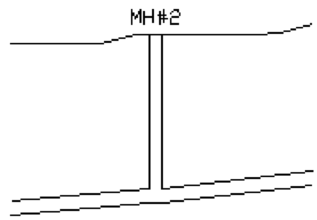
Example of sewer profile using Horizontal Axis Text Orientation as Vertical and Pipe Label Position as Horizontal Dimension

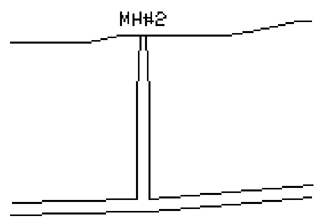




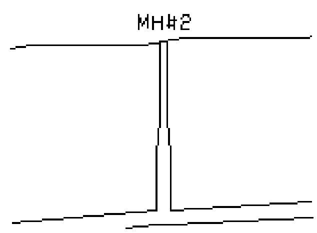
Detail of manhole bottom at pipe slope
Detail of drop across manhole of 0.2
Detail of step up

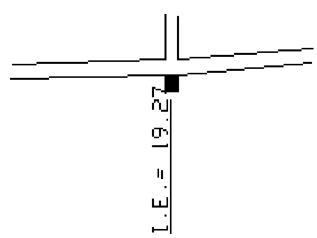


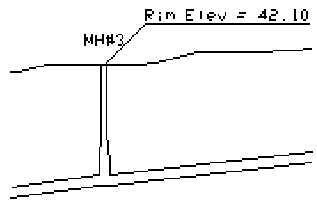




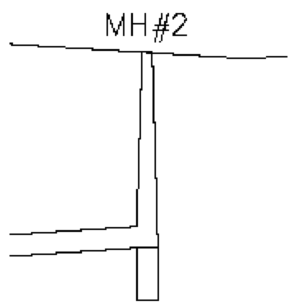
Top=2, Bottom=4, Offset=4, Fixed=0
Top=2, Bottom=4, Offset=100
Top=4, Bottom=4

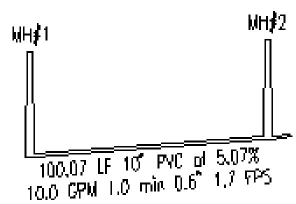






Detail of Label Rim Elevation at Manhole
Top=2, Bottom=4, Offset=4, Fixed=2
Detail of Draw Manhole Base and Label Invert Elevation with Vertical Line



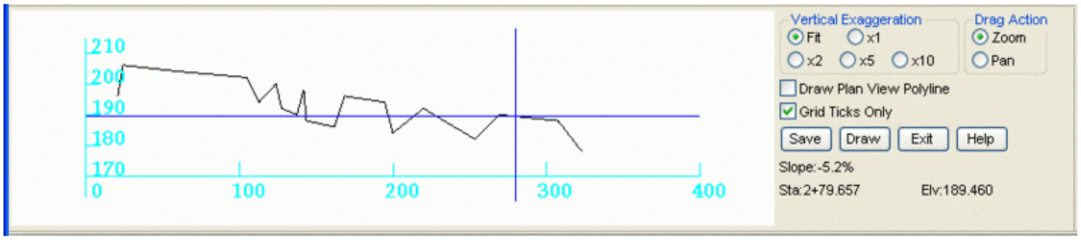


Label Pipe Flow Values option shows flow rate, travel time, depth and velocity
Manhole with the Draw Sump option

Quick Profile from Screen Entities

Function

This command allows you to create, view, and draw profiles from the current surface.



Pick starting point (CL-Centerline, P-Polyline): To make a profile you need to define the alignment by: 1) picking points on the screen; 2) typing in CL in the command prompt, and selecting a centerline file; or 3) typing in P and choosing a polyline from the screen. After doing so, the above profile viewer is created.

Quick Profile from Screen Entities creates a profile from all the entities that cross your alignment. When you move the cursor around the profile viewer a crosshair follows along the surface and reports the Station, Slope %, and Elevation at each point. It is displayed towards the bottom-right side of the screen. In this example the station is 2+79.657, the Slope is -5.2%, and the Elevation is 819.460. A crosshair can been seen in the profile drawing and along the alignment in the main drawing as well.

Vertical Exaggeration: x1 is the actual appearance of the surface(s). Depending on the flatness of the surface(s), you can select x2, x5, x10 vertical exaggerations to better see the elevation differentiation and different surfaces. The option Fit automatically exaggerates the vertical to best fit the profile viewer.

Drag Action: This dialog allows you to zoom in and out, and pan around the profile. To zoom in click and drag up, to zoom out click and drag down. To Pan, click amd drag the direction you want to move.

If you created a profile alignment by picking points and you want to save that polyline you created then toggle on Draw Plan View Polyline. If you do not choose Draw Plan View Polyline than the polyline will be lost when you exit out of the Quick Profile from Screen Enities command. Grid Ticks Only marks elevations and distances but does not draw them into grids.

The Save icon allows you to save the profile as a (.pro) file by whatever name you give it. The Draw icon allows you to draw the profile right on your drawing. Set the layer name, vertical and horizontal scale as desired, pick a starting point to draw, and the profile is created.

Prompts

Command: *tk_quickpro2*

Pick starting point (CL-Centerline,P-Polyline): *P to select a polyline from the screen*

Select profile centerline polyline: *Select the desired polyline*
Tested 39 of 39 Entities Intersects found> 12

Opening file C:/Program Files/Carlson TakeOff 2004/quickpro.pro for write.

Prerequisite: entities

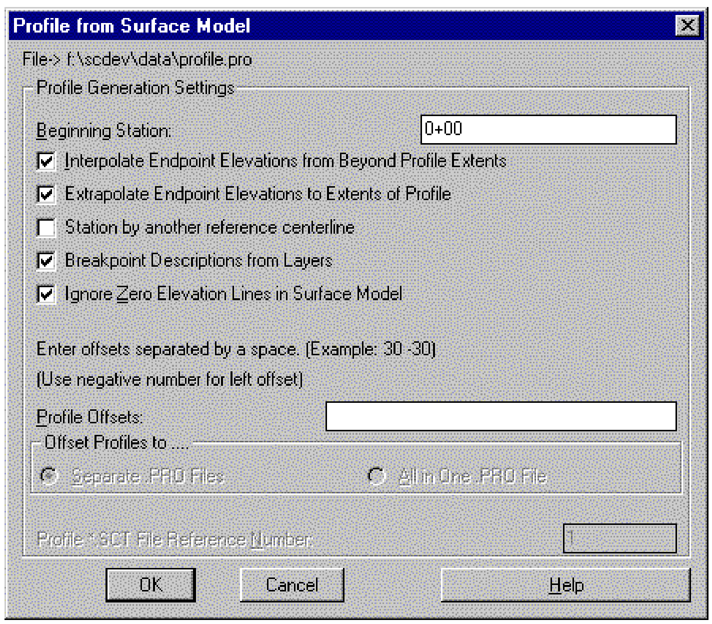
Keyboard Command: tk_quickpro2

Profile from Screen Entities

Function

Profile from Surface Entities creates a profile from contours, triangular mesh, and other 3D drawing entities. The method is to draw a polyline as the profile centerline. Then the profile is derived from the intersections of this polyline with the 3D entities. For added accuracy in pulling the profile, include the triangular mesh as well as the contours.

The Interpolate Endpoint Elevations from Beyond Profile Extents option will cause the program to look past the ends of the centerline for additional intersections with 3D entities.These additional intersections will then be used to interpolate the elevation at the starting and ending station of the centerline. The Station by Another Reference Centerline option will prompt you to pick another centerline polyline. The intersection points along the first centerline are then projected onto the second centerline. The profile then stores the elevation of the intersection with the station along the second centerline.



In addition to creating the one centerline profile, offset profiles may also be created by entering the offset distances in the Profiles Offset box of the Profile from Surface Model dialog.

Prompts

File Selection Dialog

Specify a profile file name to create.

Profile from Surface Model dialog box (previous page)

Polyline should be drawn in direction of increasing stations.

CL File/<select polyline which represents the profile centerline>: *Pick the centerline*

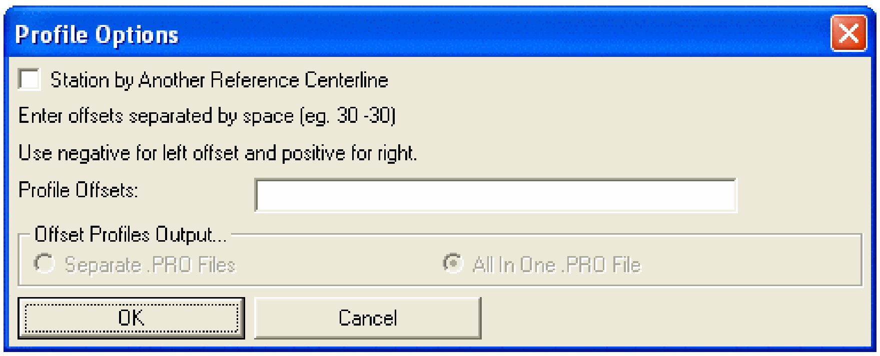
Select Lines, PLines, and/or 3DFaces that define the surface for profiling.

Select objects: *C* (for crossing and window everything the centerline crosses)

Prerequisite: A polyline centerline and surface lines and polylines.

Profile from TIN or Grid

This command creates a profile (.PRO file) from a centerline polyline and a surface model stored in a 3D grid file (.GRD) or triangulation file (.FLT). The polyline defines the alignment of the profile and the grid defines the surface.



Prompts

Choose Grid or Triangulation file to process Select existing .GRD, .TIN, or .FLT file.
Complete the Profile Options dialog.

If you choose to station by another reference centerline, it is necessary that the reference centerline extend beyond the range of the picked polyline in order to project correctly and capture offsets along the entire length of the picked centerline.

Choose PROfile file to Write dialog Enter a profile file (.PRO) name to write.
Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>: *select a polyline*
Polyline should have been drawn in direction of increasing stations.
CL File/<Select Reference centerline polyline>: *select a polyline*
CL File/<Select Reference centerline polyline>: *press Enter*
Reference CL starting station <0.0>: *press enter*

Pulldown Menu Location: Profiles > Profile from ...
Keyboard Command: progrid
Prerequisite: A .GRD grid file, .TIN, or .FLT tmesh file
File Names: \lsp\progrid.lsp, \lsp\profedit.arx

Profile from 2D Polyline

Function

This command allows you to convert a polyline that is drawn on a profile grid into a profile (.PRO) file. The polyline must be drawn in the direction of stationing.

Prompts

New or Append Dialog Box

Choose New unless you intend to create a multiple profile.

File Selection Dialog Box

Specify the profile (.PRO) file to create.

Profile Settings Dialog

Set these parameters to match the dimensions of the grid.

Pick the lower left grid corner: *Pick the grid corner.* Endpoint snap is set on.

Profile number <1>: *Press Enter.* This is an optional profile name used for multiple profiles.

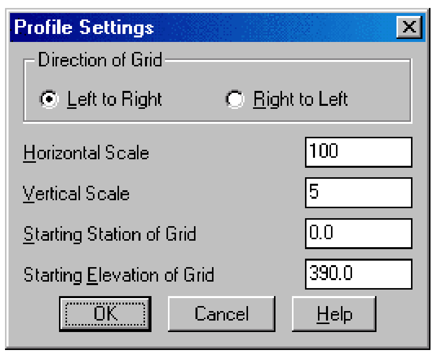
Select the polyline to write profile from:

Select object: *Pick the polyline in the grid.*

A station and elevation report is produced.

Prerequisite: Drawn polyline which represents profile.

Keyboard Command: pro2dpl



Profile from 3D Polyline

To create a .PRO file, Profile from 3D Polyline uses X-Y distances between the points of a 3D polyline for sequential stations and the Z values at these points for profile elevations.

Prompts

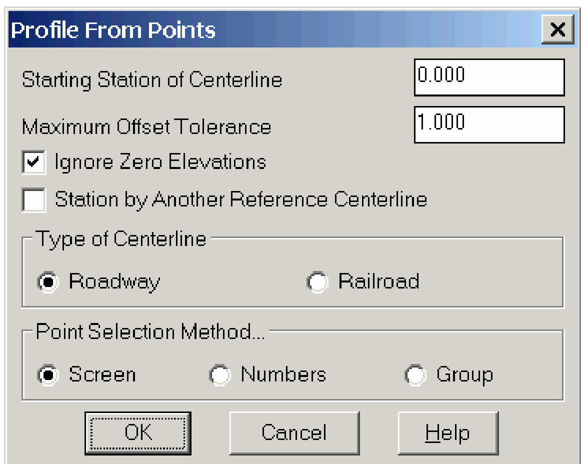
[nea on] Select polyline to Station/measure: *pick a 3D polyline*
Profile File to Write dialog Specify a profile file name to create or append an existing file.
Profile number <1>: *press Enter* This is an optional profile name useful for multiple profiles.
Station by another reference centerline [Yes/<No>]? *press Enter*
Starting Station <0.0>: *press Enter* This is the station at the start of the picked or reference polyline.

Prompt for elevations [Yes/<No>]? *press Enter*
The new profile is then stored.

Pulldown Menu Location: Profiles
Keyboard Command: pro3dpl
Prerequisite: A 3D polyline
File Name: \lsp\profilp.lsp

Profile from Points on Centerline

This command creates a .PRO file from points and a centerline that is represented by a polyline or centerline file. The elevations of the profile are derived from the elevation of the points and the stationing for these profile points is calculated from the distance along the centerline. The points must be within the offset distance from the polyline in order to be included in the profile. The profile is created by projecting the points perpendicular onto the alignment to determine the station and the elevation comes from the point elevation. The polyline or centerline should be drawn (or defined) in the direction of increasing stations. The points can be selected from point entities in the drawing (Screen), by point numbers from the current coordinate file (Numbers), or by point group as defined by the Point Group Manager (Group).



Prompts

PROfile file to Write dialog box: Enter a new profile file name to write.
CL File/<Select polyline that represents centerline>: *pick a polyline or choose C for Centerline*
Select Centerline file if Centerline option is used. If the desired points are further from the centerline, enter a larger maximum offset tolerance.
Note: for all selected points, the points should be located on the real Z axis.
Select the Carlson points along the centerline.
Select objects: Select the point entities.

Keyboard Command: profpts
Prerequisite: A polyline centerline and points
File Names: \lsp\profpts.lsp, \lsp\regrade.arx

Import Profile

Function

This command converts Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, and Leica road files into Carlson TakeOff profile (.PRO) files.

Prerequisite: a Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, or Leica road file

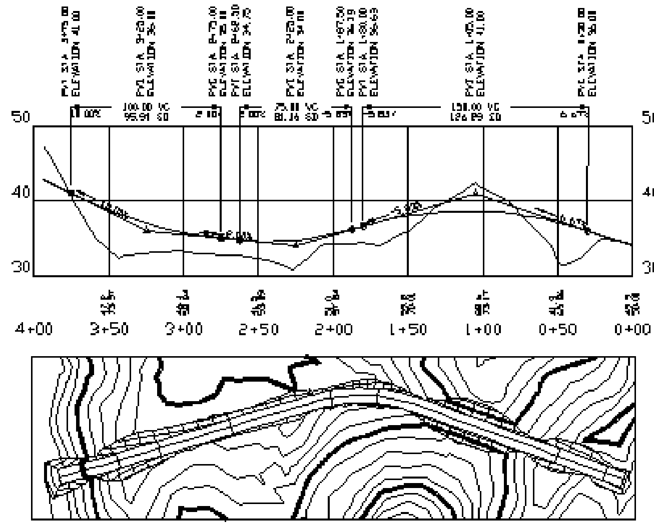
Profile To 3D Polyline

This command converts a 2D polyline centerline into a 3D polyline that follows the elevations of the profile. Horizontal and vertical curves are represented as a series of polyline segments since 3D polylines cannot contain arcs. Profile to 3D Polyline can be combined with other commands for plan-view road design as follows:

1. Draw 2D polyline centerline.
2. Profile from Surface Model - to create existing surface profile.
3. Design Road Profile - to design the final profile with vertical curves.
4. Profile to 3D Polyline - create a 3D polyline of the road centerline.
5. Offset 3D Polyline - offset the 3D polyline centerline left and right by the horizontal and vertical distances.
6. Design Pad Template - run twice for left and right polylines of road to tie into surface at specified cut and fill slopes. This creates the limits of the disturbed area. Or use Join Nearest, Direct Connect Endpoints, to create a closed loop pad with one run of Design Pad Template for simple ramps, driveways and access roads.
7. Triangulate & Contour - draw final contours using road 3D polylines.
8. Volumes - use any of the volumes commands to calculate cut and fill volumes.

Prompts

Layer Name for 3D Polyline <3DPROF>: *press Enter*
Select profile centerline polyline: *pick a polyline*
Station by another reference centerline [Yes/<No>]? *N* for no. This option will prompt for a second centerline to use for stationing.
Enter the starting station <0.0>: *press Enter*
Select Profile File
Starting station of centerline <0.0>: *press Enter*
Erase centerline (Yes/<No>)? *Y* This option will erase the original 2D polyline centerline.

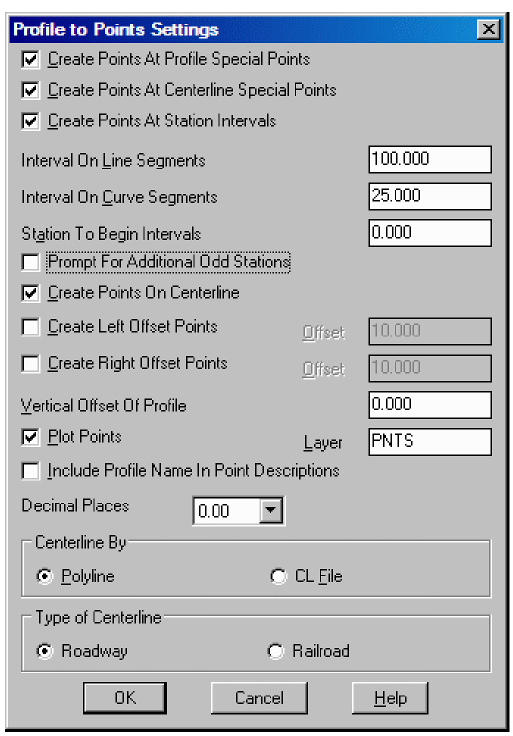


Example of road design in plan-view with Profile to 3D Polyline

Keyboard Command: proto3dp
Prerequisite: A .PRO file and a centerline polyline
File Name: \lsp\profedit.arx

Profile To Points

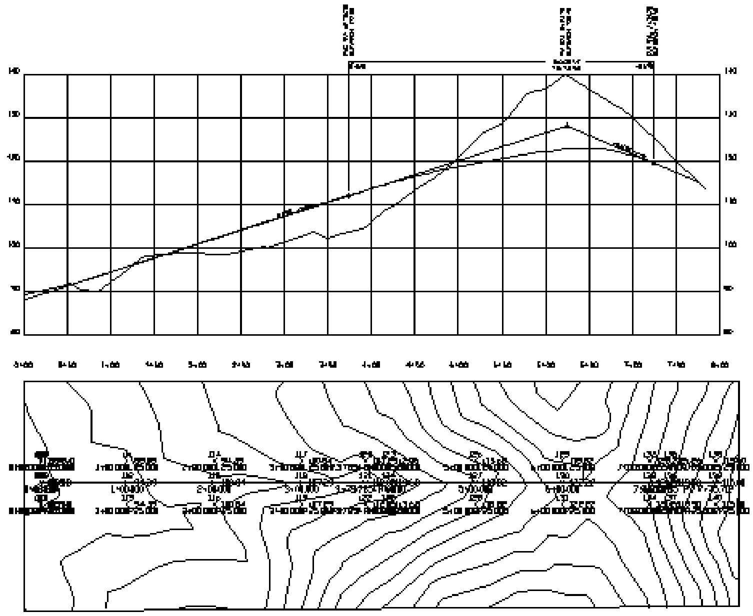
This command creates Carlson points along a horizontal alignment polyline using a profile file to compute the point elevations. The created points are stored in a coordinate (.CRD) file and can also be drawn on screen in the layer specified by the user. Station text, profile name, and special points (vertical and horizontal PC's and PT's) can be stored in the point description depending on user settings.



Create points at Profile special points: Includes vertical PC and PT points.
Create points at Centerline special points: Includes horizontal PC and PT points.
Create points at Station Intervals: Allows you to specify intervals for point creation.
Interval On Line Segments: Specify station interval for line segments.
Interval On Curve Segments: Specify station interval for curve segments.
Station to Begin Intervals: Specify station to start intervals.
Prompt For Additional Odd Stations: Any station can be entered to create additional points with elevations derived from the profile.
Create Points on Centerline: When checked, points will be created on the centerline.
Create Left Offset Points: When checked, left offset points will be created. Specify the offset in the edit box.
Create Right Offset Points: When checked, right offset points will be created. Specify the offset in the edit box.
Vertical Offset of Profile: Specify the vertical offset. Enter zero for no vertical offset.
Plot Points: When checked, points will be plotted in the drawing, otherwise points are only added to the current coordinate (.CRD) file.
Include profile name in point descriptions: When checked, the profile name will be used as the prefix on the the point description. For example, if the profile name is DESIGN.PRO, then the point description might be DESIGN 0+63.37.
Decimal Places: Specify the display precision for points that are plotted in the drawing. This setting does not affect the coordinates stored in the CRD file.
Centerline by: Click either Polyline or CL File.
Type of Centerline: Click either Roadway or Railroad.
OK: Specify files.

Prompts

Select Coordinate File to Process
If the current coordinate is set, it is used automatically without this prompt.
Select profile centerline polyline: *pick a polyline*
Station by another reference centerline [Yes/<No>]? *N for no*. This option will prompt for a second centerline to use for stationing.
Starting station of centerline <0.0>: *press Enter*
Choose Profile to Process dialog Specify a profile name.
Starting point number <1>: *press Enter* This defaults to the point number after the highest one currently in the CRD file.
Station for additional point (ENTER to end): *press Enter* This option will create a point at the specified station. Prompt occurs only if option is specified in dialog.



Points created along profile centerline using elevations from the above road profile

Keyboard Command: pro2pts
Prerequisite: A .PRO file and a centerline polyline
File Name: \lsp\profedit.arx

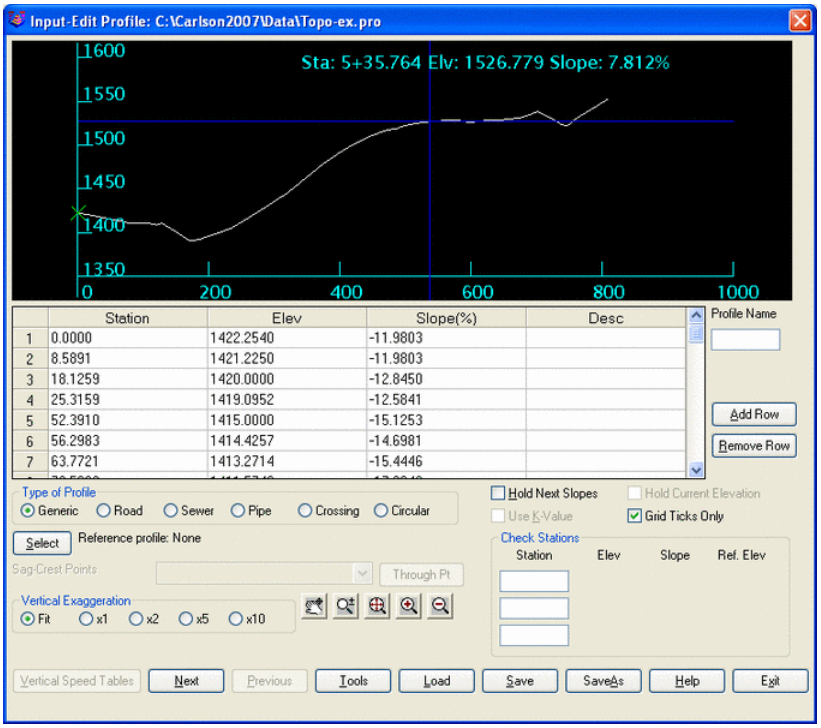
Input-Edit Profile File

This command is a spreadsheet type editor for profile (.PRO) files. Besides editing a profile, this routine can be used to just view the contents of a profile. Also, a new profile can be entered by editing a previously empty or non existing file.

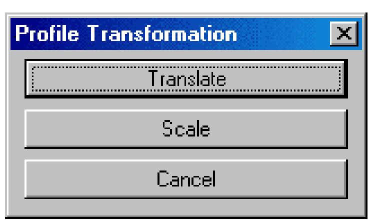
The command starts by prompting for the profile file to edit. Alternately, you can run Input-Edit Profile by double-clicking on a profile polyline that is drawn on a profile grid.

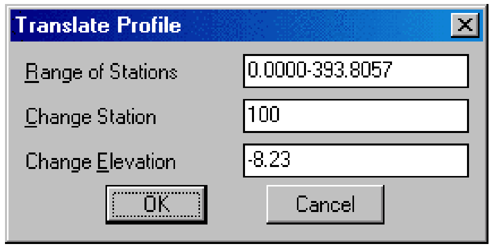
The opening dialog below shows the layout of this editor. At the top of the dialog, you can dynamically see the profile and vary its appearance by using zoom and pan. You can change the look of the profile more by using the vertical exaggeration multipliers. The station, elevation and slopes are also shown at the lower left of the dialog, fluctuating with the movement of the cursor. Then there are between five and nine columns for the possible fields in a profile. Which columns are active depends on the type of profile: generic, road, sewer, pipe, crossing or circular. Six rows are visible at a time. To view different rows, use the scroll bar on the right. When a greater amount of columns are in use, use the scroll bar at the bottom. The Profile Name edit box is an optional identification name used by multiple profiles in Draw Profile. The Add Row and Remove Row buttons, when used, will dynamically and immediately make changes to the profile image at the top.

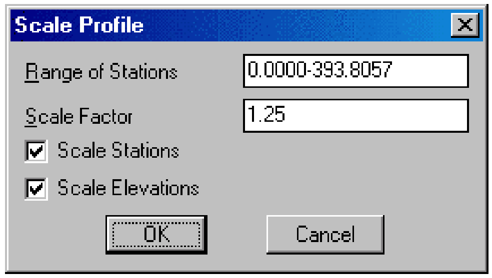
On the right is a column for Check Stations which report the elevation at the specified stations. The Check Stations are not stored in the profile. This is a design tool for viewing the elevations at certain stations while adjusting the profile data. The last line has eight action buttons.



Add Row: Adds a new row into the profile after the current row.
Remove Row: Removes the current row.
Type of Profile: Choose. Column titles and the amount of columns will change accordingly.
Hold Next Slopes: A toggle that may applied or left blank.
Use K-Value: Toggles between displaying K-Value and Sight Distance in the fifth column for road profiles.
Select Reference profile: An option to show a second profile as reference. When a reference profile is active, the Check Stations and graphic window report the cut/fill with the current profile and the elevation of the reference profile. Also with a reference profile active, the spreadsheet adds a column for depth.
Vertical Speed Tables: This button is enabled only when you edit a road profile. Please refer to the documentation on Input-Edit Road Profile for the information on Vertical Speed Tables.
Next: Used for navigation when editing a .PRO file containing multiple profiles, loads the next profile.
Previous: Used for navigation when editing a .PRO file containing multiple profiles, loads the previous profile.
Transform: Allows you to either Translate or Scale the profile. Translate globally adds or subtracts value to stations and/or elevations within the specified range of stations, while Scale will apply the specified scale factor to stations and/or elevations within the specified range of stations.
Load: Used for loading another, existing .PRO file for editing.
Save: Saves the profile using the current profile file name. The current profile file name is displayed in the top title bar of the dialog box.
SaveAs: Allows you to save the profile under a different profile file name.



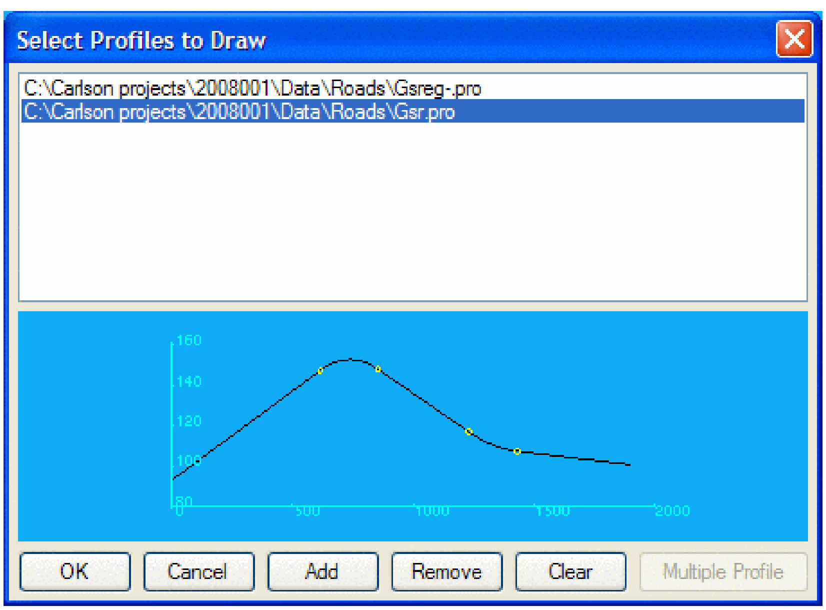




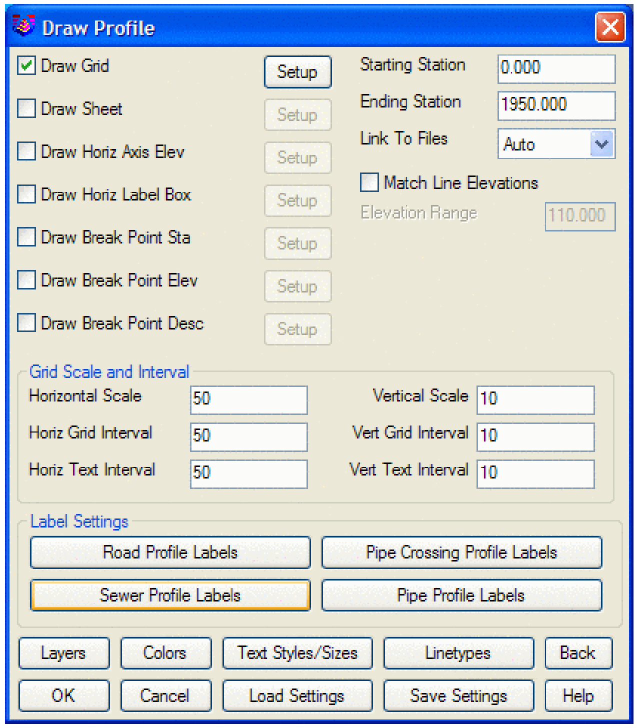
Keyboard Command: profedit
Prerequisite: None
File Names: \lsp\profile.dcl, \lsp\profedit.arx

Draw Profile

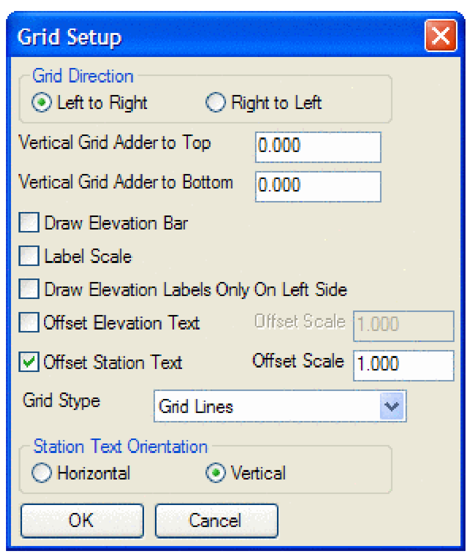
Draw Profile is a flexible routine for drawing a profile anywhere in the drawing. The profile may be drawn with or without a grid or with just tick marks. The vertical curve annotations, for a road profile, and manhole annotations, for a sewer profile, may also be drawn. Draw Profile uses the profile information that is stored in .PRO files. Once the profile is drawn using Draw Profile, the design and labeling routines of the Profiles dropdown are applicable to the profile. The first step in Draw Profile is to choose the profile (.PRO) file(s) you want to draw.



The Draw Profile dialog box appears, and contains all of the settings for creating the profile.



Draw Grid: This option will draw a grid and axis elevations for the profile. Pick Setup to access Grid Setup dialog.



Grid Direction: Profiles can be drawn Left to Right (the default) or Right to Left. Although most profiles are drawn left to right, if you have a road that runs east to west and you wish to draw the profile stationing beneath the actual road stationing, then choosing a Right to Left profile may be appropriate. Unavailable when Draw Sheet is checked.

Vertical Grid Adder to Top: This adds the specified amount of grid to the top of the profile.

Vertical Grid Adder to Bottom: This adds the specified amount of grid to the bottom of the profile.

Draw Elevation Bar: Click on this option if you desire to have a vertical barscale displayed. It will run up and along the left-most vertical grid line of the profile.

Label Scale: Click on this option and you obtain a scale drawn at the lower left corner of the profile.

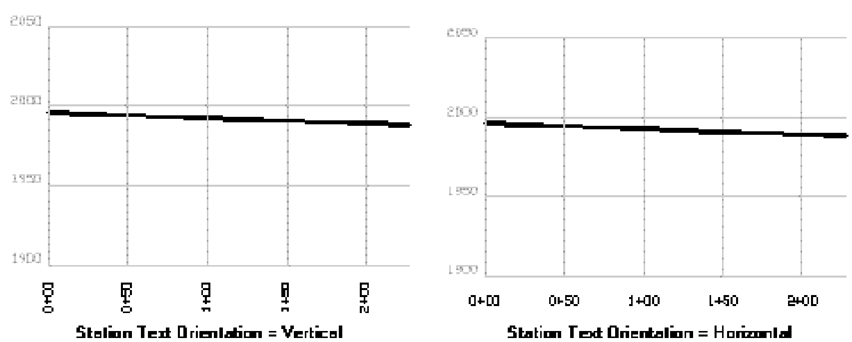
Draw Elevation Labels Only On Left Side: This option eliminates elevation labels on the right side of the profile.

Offset Elevation Text: This option offsets the left-side vertical axis text using the specified Offset Scale.

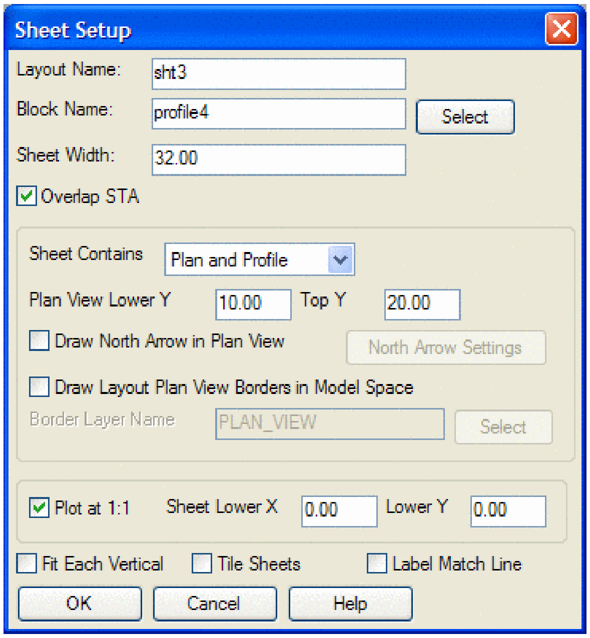
Offset Station Text: This option offsets the horizontal axis Station text by the specified Offset Scale, allowing the insertion of elevation or other information above the stationing. It is often used in conjunction with the Label Horizontal Axis options.

Grid Type: This selects the type of Grid to generate. The choices are Grid Lines, Ticks Only, Ticks and Dots, Ticks and Checks.

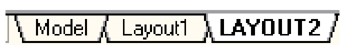
Station Text Orientation: This option allows you to specify the orientation of the station text shown along the bottom of the profile. The example below shows both options:



Draw Sheet: When checked, the profiles will be drawn in paper space. Plan Only, Profile Only, or Plan and Profile sheets can be created. Each plan and profile sheet is created in it's own layout tab. When the plan and profile is drawn, you are placed in tilemode=0 and paper space. Click the "model" space tab (shown below) to return to model space to edit the plan view features, for example. The options within Sheet Setup become available when this toggle is checked on. Pick Setup to access the Sheet Setup dialog:



Layout Name: Enter a name for the paper space "tabs" to be assigned to each layout for each sheet. The program will automatically divide the plan view and the profile view into sheet layouts, and if the length of the profile extends beyond a single sheet, then multiple layouts are created, with the layout name ID incremented by 1. If you enter "ms" to go to model space within a layout tab, you can pan to alter the plan view position. Its best to zoom in/out and edit within the Model tab. The Layout tabs appear at the bottom of the screen, along with the "Model space" tab to go back to standard plan view:



Block Name: This is the drawing name for the plan and profile sheet to be inserted. Carlson provides a standard plan and profile drawing in the form of Profile.dwg located in the Support subdirectory. You may wish to revise Profile.dwg, add your company logo, and re-save it as Profile1.dwg or you could add your own complete version of a Plan and Profile sheet. If you choose the latter, you should examine the scale, dimensions and the lower left corner of Profile.dwg, and try to duplicate those dimensions and corner coordinates in your own drawing. It is also important to store all your standard profile sheets in the Support subdirectory. You cannot draw Right to Left in Sheet mode. Note that the Sheet mode will re-orient the centerline left to right, which may cause text (such as the stationing) to plot upside down, until you use the *Flip Text* command under the Edit menu.

Sheet Width: This is the profile width, in inches, on the sheet. Even though the sheet is a fixed size, you can limit the length of the plot to 32 inches or less with this entry. If we used an entry of 16 for the example profile above, two plan and profile sheets would be created, because the first sheet would go from station 0+00 to 3+20 (16 inches at a 20 scale) and the second sheet would finish from 3+20 to 4+51.91. In English units, a typical entry here would be 30 for 30 inches.

Overlap Station: In multiple plan and profile sheet plotting, after the first sheet, all subsequent sheets will have the first 2 stations in common with the last 2 stations on the previous sheet, if the Overlap Station option is turned on. For example, if the last 2 stations are 3+10 and 3+20 on sheet 1, then sheet 2 will start with 3+10, then 3+20, with this option turned on. With this option turned off, if the first sheet ends with 3+20, then the second sheet would begin with 3+20.

Sheet Contains: This drop list allows the selection of which type of sheet to generate. The choices are Plan and Profile, Plan Only or Profile Only.

Plan View Lower Y: This sets the lower position of the paper space window for the plan view. With Lower Y set to 9 (inches above the base of the sheet) and Top Y set to 21, there is a 12 inch vertical window, running the full Sheet Width (typically 30 to 32). This window for the plan view can be expanded or reduced with these settings.

Top Y: This sets the top vertical limit for the plan view window, measured in inches from the bottom of the plan and profile sheet.

Draw North Arrow in Plan View: This draws a North Arrow in plan view.

Draw Layout Plan View Borders in Model Space: This draws the borders in Model Space.

Plot at 1:1: With this clicked on, the sheet will be paper size, designed to be plotted at 1:1. A 30-inch profile sheet will measure 30 units, even though the centerline and profile may be 1500 feet in length. If the Scale 1:1 option is turned on, then you cannot check the distances of features using commands such as Bearing and Distance on the Inquiry menu, because the distances will be scaled down by a factor equal to the drawing scale (for example, at 1"=50', the reduction in scale factor is 1/50 or 0.02). You can set the absolute starting coordinate for the 1:1 scaled plot by setting the Sheet Lower X and Sheet Lower Y. With this clicked off, the profile will drawn full size, with a 1500-foot profile measuring 1500 feet.

Fit Each Vertical: With this option turned on, the command will recognize the lower and upper vertical elevations of the profile and set the vertical axis elevation range to enclose the actual elevation limits of the profile. With this option turned off, you can enter the lower vertical elevation range, dropping it down further to increase the lower margin. Whether or not the Fit Each Vertical option is turned on or off, you are always prompted for the top elevation range.

Tile Sheets: If clicked on, only one Layout is created in paper space, and all sheets appear in this single Layout as tiles of individual sheets, much like the tiles mode of viewing files within Windows Explorer.

Label Match Line: When clicked on and multiple sheets are plotted with plan view option on, a match line will plot in the plan view.

Prompts (Draw Sheet option)

You are first asked to select the polyline that represents the centerline, and the program best fits the centerline in the plan view portion of the plan and profile sheet, then captures all of the associated drawing that will fit in that paper space window. If the length of the polyline divided by the scale exceeds the Sheet Width entry (for example, 5000 feet of road divided by 50 Horiz. Scale is 100, which exceeds the sheet width), then multiple plan and profile sheets will be automatically created. A 5000 foot road at 50 scale with a 30-inch sheet width, would lead to 3 full sheets of 1500 feet each and a "leftover" fourth sheet showing the last 500 feet.

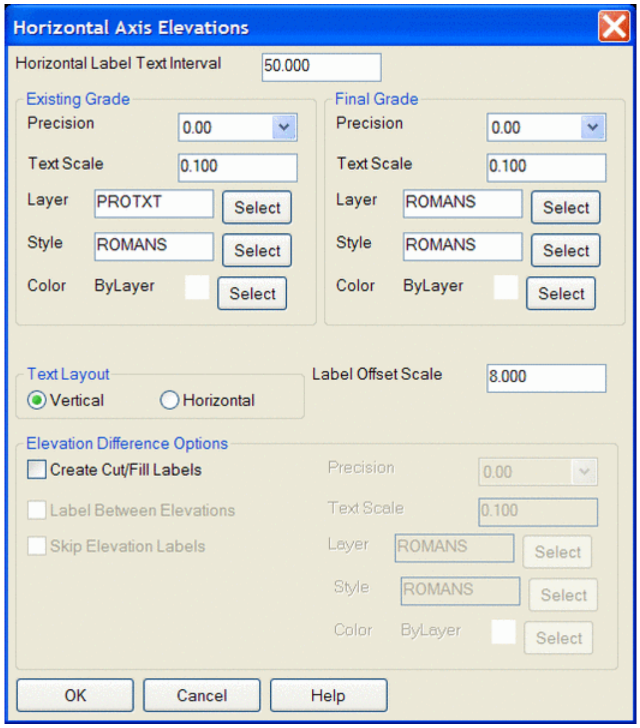
Bottom Vertical Spacing <0.0>? *press Enter*

The program places the profile plot flush against the bottom of the vertical grid, by default. This prompt allows for an offset, moving the profile plot up off the bottom of the grid. If the lowest elevation of the profile is 940 by default, entering 10 would start vertical axis labeling at 930, and if the vertical scale was 5 units, this would push the vertical plot up 10 units or 2 standard grid intervals.

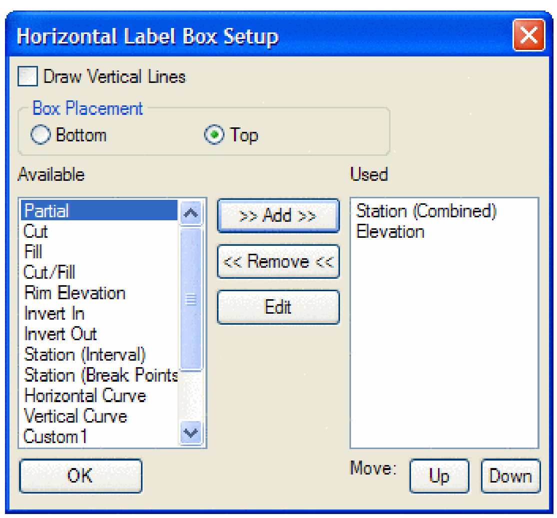
Top Elevation of Profile Grid <945.0>: *press Enter*
Cl File/Select polyline that represents centerline: Select the centerline polyline (if Draw Plan has been clicked on).
Beginning Station <0.0>: *press Enter*

The items below refer again to the profile options that are independent of the Draw Sheet option:

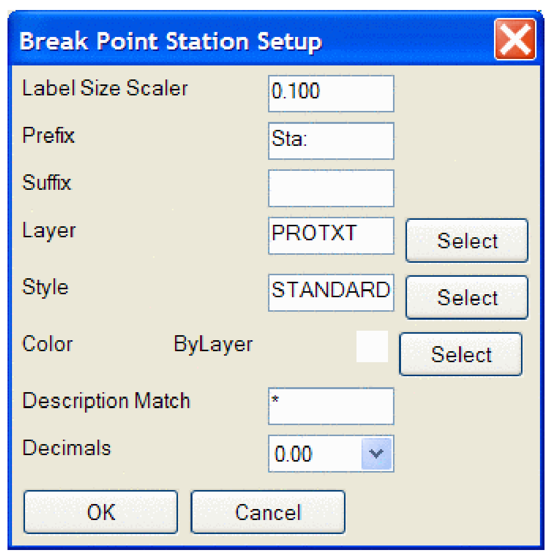
Draw Horiz Axis Elev: This option creates elevation labels along the horizontal axis. Pick Setup to access the Horizontal Axis Elevations settings dialog.



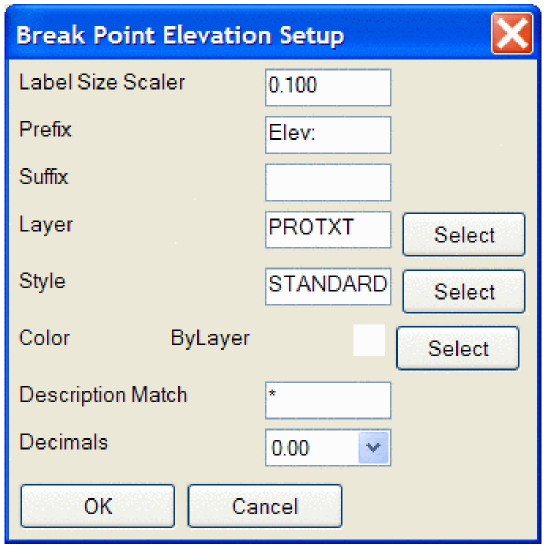
Draw Horiz Label Box: This option draws a boxed area underneath the profile. It is best used in standard Draw Grid mode, with Draw Sheets clicked off. Pick Setup to access the Horizontal Label Box Setup dialog. An example of the resulting plot is shown here:



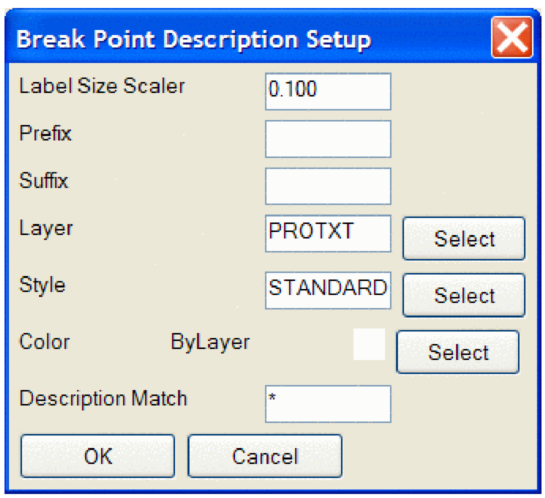
Draw Break Point Sta: Will label these values along the profile line above each break point in the profile. Pick Setup to access the Break Point Station Setup dialog.



Draw Break Point Elev: Will label these values along the profile line above each break point in the profile. Pick Setup to access the Break Point Elevation Setup dialog.



Draw Break Point Desc: Will label these values along the profile line above each break point in the profile. Pick Setup to access the Break Point Description Setup dialog.



Starting Station: This field defaults to the starting station in the selected profile(s). If changed, the starting station can move forward, clipping out the first part of the profile. When you are not plotting sheets, you must set the starting station to the end of the previous sheet's ending station to force a multiple sheet layout.

Ending Station: This field defaults to the ending station in the selected profile(s). A profile that is 3000 feet in length could be plotted in 2 parts, first station 0 to 1500, then station 1500 to 3000, using the Starting Station and Ending Station options.

Label Text Scaler: This sets the size of text used for vertical curve annotation to the horizontal scale times the scaler, when you are working in English units. In metric units the text height would be 0.01∗horizontal scale∗scaler.

Link To Files: This setting controls the linkage of the plotted profile(s) to the actual profile file(s) (.PRO), determining how changes to the file affect the plotted profile(s). If set to Off, there is no linkage, Prompt will

ask whether to update the plotted profile(s) when the file changes, and Auto will automatically update the plotted profile(s) when the file changes.

Match Line Elevations: For high relief profiles that might otherwise extend up and into the plan view portion of the drawing, the Match Line Elevations option can be used to break the profile and redraw the remaining portion with its own vertical scale, as seen above.

Elevation Range: This is the range of elevations that is used in conjunction with the Match Line Elevation option. If the range is exceeded (that is, if the range above is 20), the program will break the profile and draw the remainder with a separate vertical axis range.

Grid Scale and Interval Settings

Horizontal Scale: This scale applies primarily to text size. If the text scaler is 0.1 and the horizontal scale is 50, then text size will be $0.1 * 50 = 5$.

Horizontal Grid Interval: This sets the spacing of the grids that run vertically from the horizontal scale.

Horizontal Text Interval: This sets the spacing of the stationing text that appears along the horizontal axis. When using a large "Axis Text Scaler", the horizontal axis text can become too large, and it often necessary to space the horizontal text interval at twice the horizontal scale.

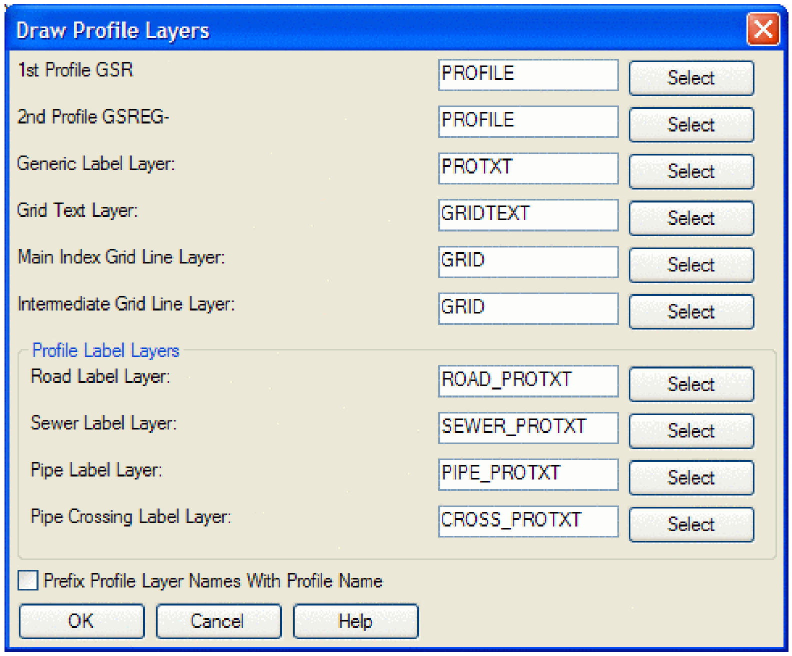
Vertical Scale: This scale sets the vertical exaggeration of the profile. If the horizontal scale and vertical scale are the same, then the vertical is not exaggerated. Profiles are often plotted with a 5 or 10 vertical exaggeration. For example, the horizontal scale may be 50, but the vertical scale may be 5.

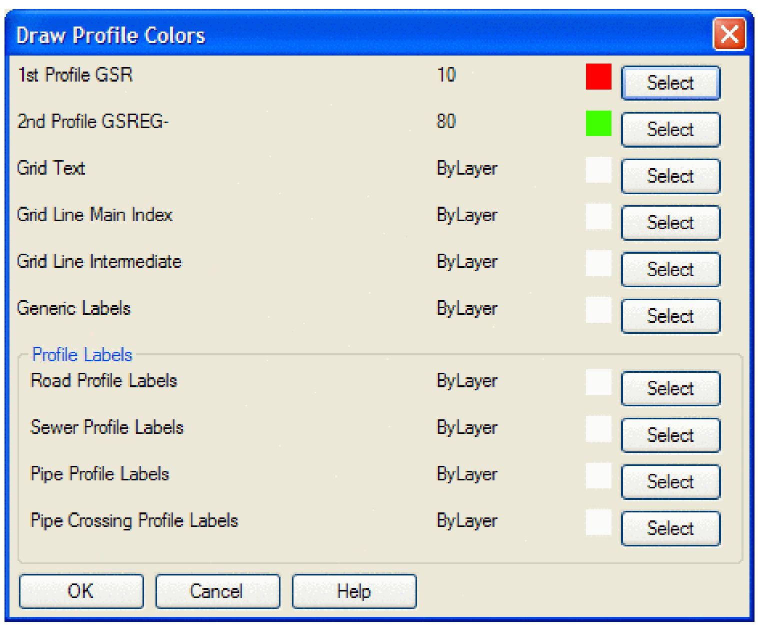
Vertical Grid Interval: This sets the spacing of the grids that run horizontally between the vertical axes on the left and right side of the profile.

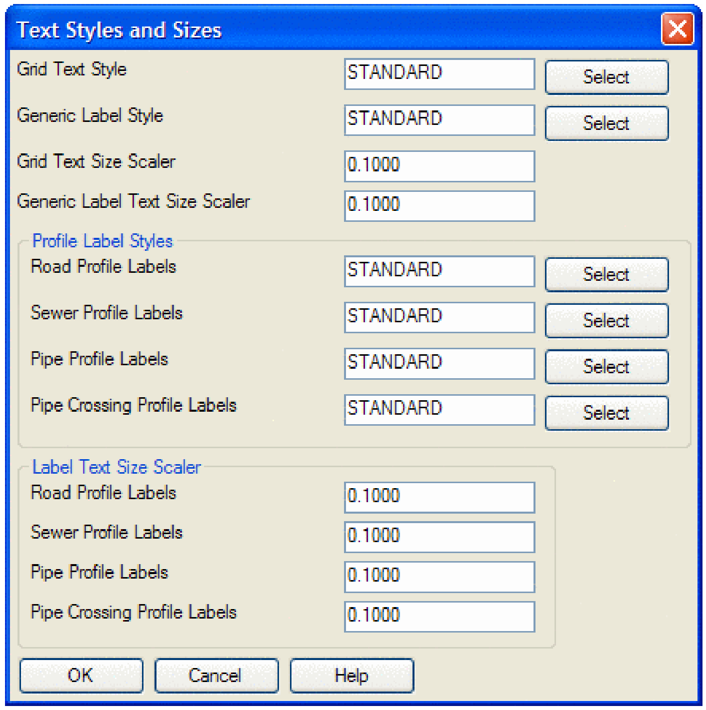
Vertical Text Interval: This sets the spacing of the elevation text that appears along the vertical axes.

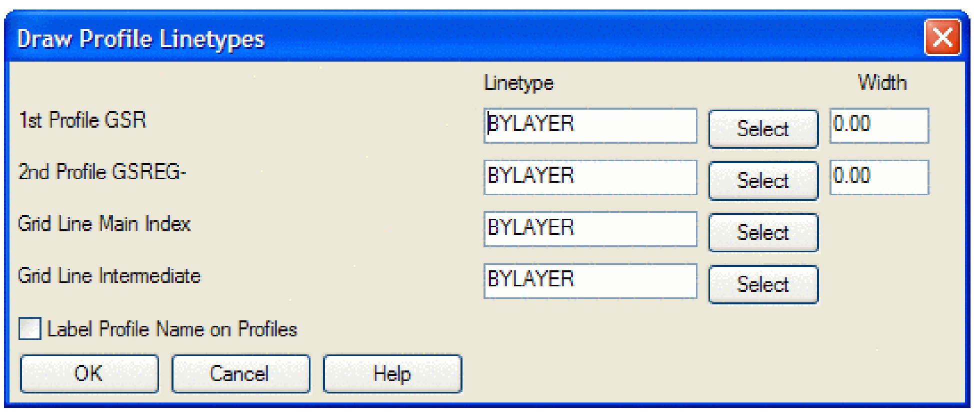
Label Settings: These 4 buttons are where you gain access to control over specific label settings for different profile types.

Layers, Colors, Text Styles and Linetypes buttons provide access to settings for each of these features of the profiles.


Draw Profile Layers
1st Profile GSR
PROFILE
Select
2nd Profile GSREG-
PROFILE
Select
Generic Label Layer:
PROTXT
Select
Grid Text Layer:
GRIDTEXT
Select
Main Index Grid Line Layer:
GRID
Select
Intermediate Grid Line Layer:
GRID
Select
Profile Label Layers
Road Label Layer:
ROAD_PROTXT
Select
Sewer Label Layer:
SEWER_PROTXT
Select
Pipe Label Layer:
PIPE_PROTXT
Select
Pipe Crossing Label Layer:
CROSS_PROTXT
Select
Prefix Profile Layer Names With Profile Name
OK
Cancel
Help


Draw Profile Colors
1st Profile GSR
10
Select
2nd Profile GSREG-
80
Select
Grid Text
ByLayer
Select
Grid Line Main Index
ByLayer
Select
Grid Line Intermediate
ByLayer
Select
Generic Labels
ByLayer
Select
Profile Labels
Road Profile Labels
ByLayer
Select
Sewer Profile Labels
ByLayer
Select
Pipe Profile Labels
ByLayer
Select
Pipe Crossing Profile Labels
ByLayer
Select
OK
Cancel
Help





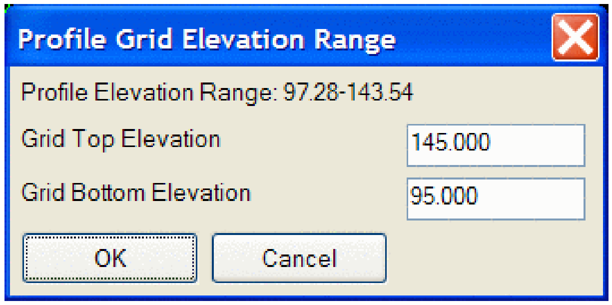
Load Settings: Loads a saved collection of Draw Profile settings, saved in a (.PFS) file.
Save Settings: Saves all Draw Profile settings in a (.PFS) file.

When OK is clicked at the base of the dialog box, the prompting at the command line continues. In this example, assume that a road profile has been selected, since more prompts will occur with road profiles than with generic profiles.

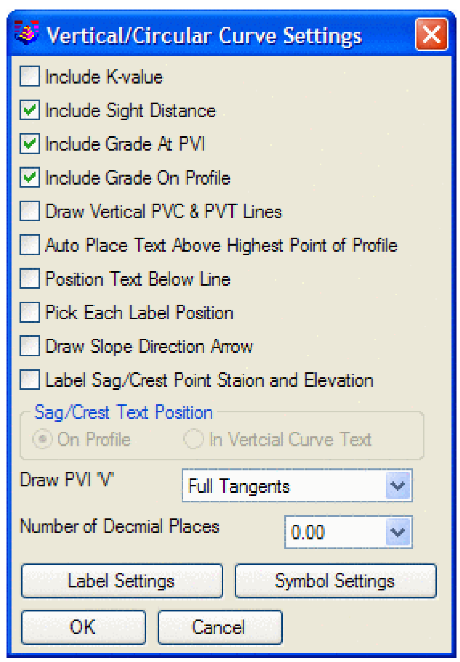
Prompts (Road profile example)

Erase existing profile from drawing [<Yes>/No]? *N* This prompt appears only if you have previously drawn the profile.

Next a dialog appears to set the Profile Grid Elevation Range by specifying the Top and Bottom Elevations. Adjust as desired, pick OK.



Pick Starting Point for Grid <8779.55 , 5716.36>: *pick a point for the lower left corner of the grid*
Assuming a road profile has been selected, the following dialog box appears:



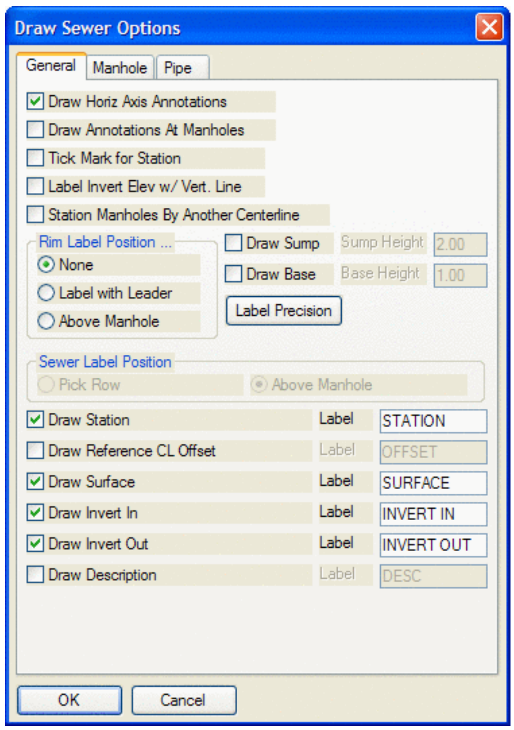
Include K-value: This option is a function of the change in slopes on either side of the point of vertical intersection.
Include Sight Distance: This option is computed by the delta slope and is a function of whether the vertical curve is a sag or a crest.
Include Grade: This option draws slopes along the tangent portion of the vertical curves, with slope direction arrows.
Draw Vertical PVC and PVT Lines: This option draws vertical lines emanating from the PVC and PVT of all vertical curves.
Position Text Below Line: This option draws the PVC, PVI, and PVT information under the picked location for the vertical curve labeling.

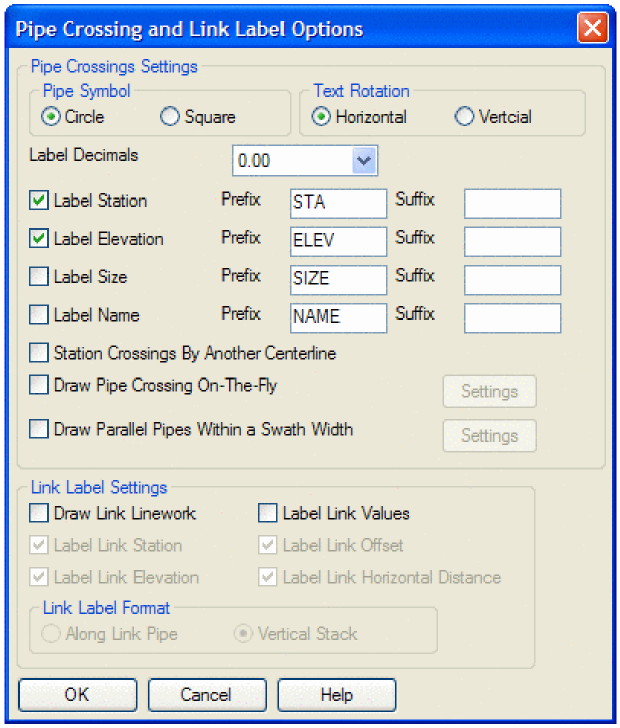
Pick Each Label Position: If there were more than one vertical curve in the profile, this option allows you to pick a vertical position for each of the vertical curve's annotation.
Draw Slope Direction Arrow: Draws an arrow to indicate slope direction.

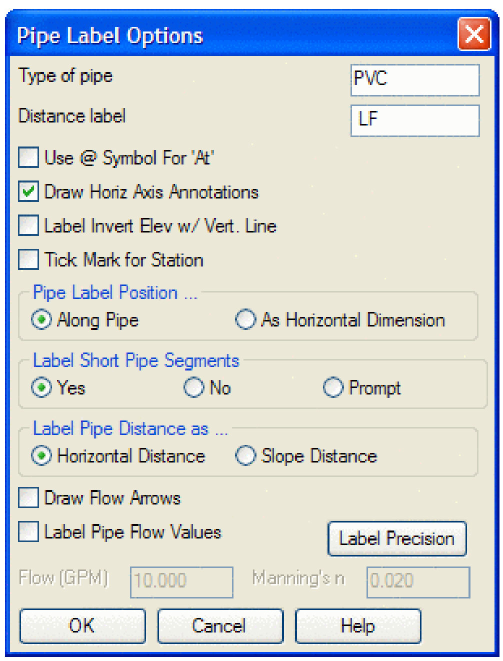
Number of Decimal Places: Choose the decimal precision used in the elevation and stationing annotation for vertical curves.

Next, select a point vertically that corresponds to the position of the left-right lines under which is written percent grade and above which is written the vertical curve length, sight distance, and K-factor, if requested. The PVC, PVI, and PVT stations and elevations are written above or below this picked point depending on dialog box settings.

Sewer Options:







Additional Prompting for Multiple Profiles

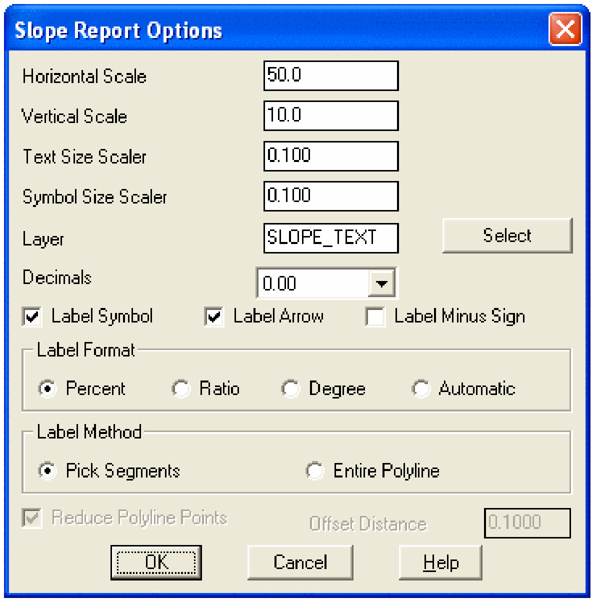
Detected multiple profiles within C:\CARLSON PROJECTS\2006-0124\DATA\PRO\EXAMPLE.PRO
Note that the *Profile from Surface Entities* command can store additional profiles into the same profile file.
Draw profiles on same or different grids (Same/<Different>)? *press Enter* This determines whether the multiple profiles will be drawn together on the same grid or drawn on separate grids.
Uniform or variable grid size (Uniform/<Variable>)?*press Enter* This selects between one-size-fits all grids or

individually sized grids for each profile.

Keyboard Command: drawprof
Prerequisite: A .PRO file
File Names: \lsp\drawprof.lsp, \lsp\profile.dcl, \lsp\vcplot.lsp, \lsp\endsewer.lsp

Profile Slope Report

This command calculates and labels the slope of a line, polyline segment, an entire polyline, or pair of points, as drawn on a profile. The command starts with the Slope Report Options dialog.

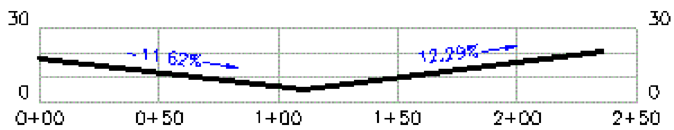


Horizontal Scale: Specify the horizontal scale of the profile.
Vertical Scale: Specify the vertical scale of the profile.
Text Size Scaler: Specify the text size scaler.
Decimals: Specify the display precision for the slope labels.
Label Symbol: When checked, the degree symbol or percent sign will be used in the label.
Label Arrow: When checked, a slope direction arrow will be included.
Label Minus Sign: Will label a minus sign on negative slopes.
Label Format: Specify how to label the profile slopes. The automatic settings means to use a percent label for any slope less than 10%. and a ratio for any slope greater that 10%.
Label Method: Choose to label the entire profile at once or to pick individual segments.
Reduce Profile Points: When checked, the number of labels created on the profile will be reduced based on the Offset Distance value. Applies only to the Entire Polyline selection option..
Offset Distance: Specify maximum offset between profile vertices. Only available when Reduce Profile Points toggle is checked on.

Prompts

Slope Report Options dialog box
Points/<Select line or polyline to list-label>: *pick a polyline*
Slope Distance> 600.33 Horizontal Distance> 600.00
Elevation Difference: 20.00 Slope Ratio: 30.00:1 Slope Percent: 3.33

Starting point of label ([Enter] for none): *pick a point*
Points/<Select line or polyline to list-label>: *press Enter* If you choose P for points, you go into the Points mode and can label the slope of any pair of screen picks on the profile.

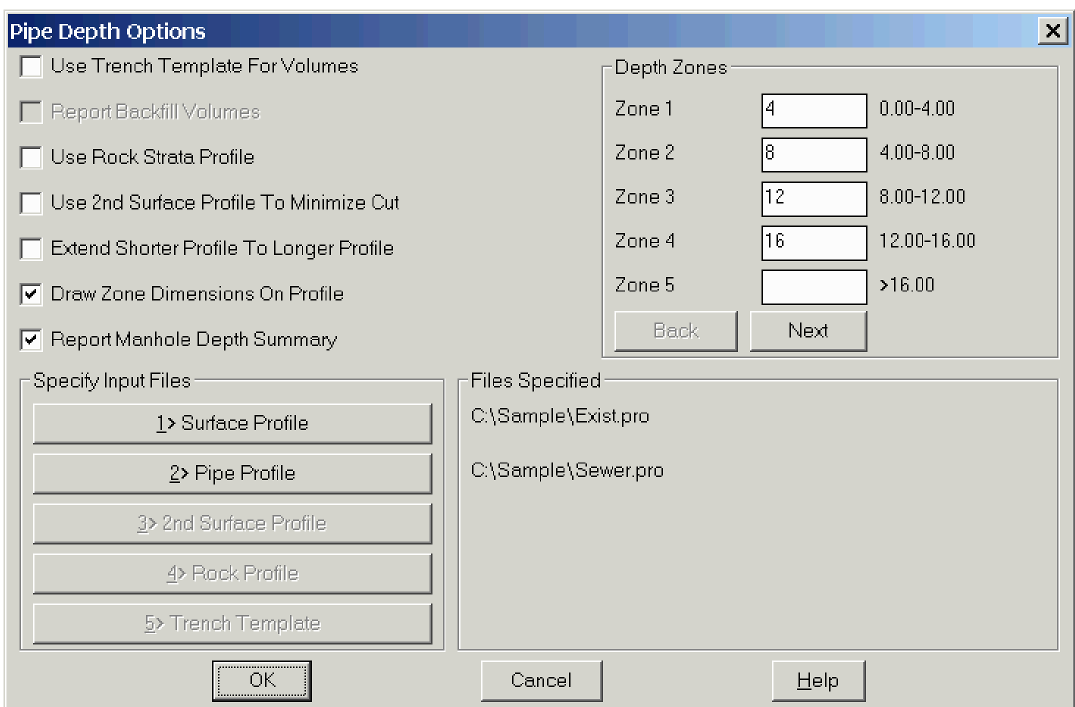


Keyboard Command: llg
Prerequisite: A profile grid and profile polyline
File Name: \lsp\llg.lsp

Pipe Depth Summary

This command reports the horizontal distances for the range of depths comparing a surface profile to a trench, pipe or sewer profile. There is an option to use two surface profiles and the program will use the minimum of the two depths. In addition to the report, the depth ranges can be labeled along the profile in the drawing.

The simplest of applications of this command, comparing a sewer profile to a surface profile and reporting the depth summary according to the specified Depth Zones, is shown below.



Use Trench Template for Volumes: Trench templates are made using the command Input-Edit Trench Template within the Profile Utilities "flyout". Trench earthwork volumes are then computed.
Report Backfill Volumes: Available if trench templates is clicked on.
Use Rock Strata Profile: If clicked on, the Rock Profile can be entered in the lower portion of the dialog, and if the pipe invert is below rock surfaces along any segment, rock linear feet will be reported, in the same depth categories

as used for trench depths. In the example shown below, if rock depth is uniformly 5 feet below surface elevation, in the form of a rock profile, rock quantities are 348 feet of 0-2 feet depth of rock trenching.

Use 2nd Surface Profile to Minimize Cut: If the final grade is below existing grade, in those areas, it saves trenching work to first do the cut to final grade, prior to filling over existing grade in areas of fill. Then trench depths are minimized. This option, if clicked on, computes trench depths to the minimum of the two specified surfaces, and activates the 2nd Surface Profile option in the lower portion of the dialog.

Extend Shorter Profile to Longer Profile: This option will extrapolate the starting and ending stations of the shorter profile to match the longer profile.

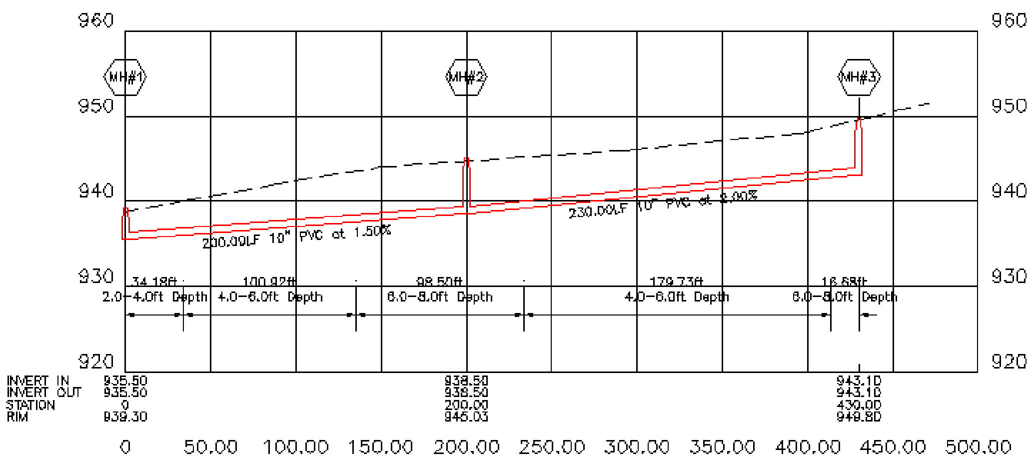
Draw Zone Dimensions on Profile: The depth zones will be annotated along the horizontal axis of a profile drawing with this option.

Report Manhole Depth Summary: This leads to the depth summary report.

Depth Zones: These zones are for reporting the pipe range of depth. The depths should be entered in lowest to highest order. Use the Next and Back buttons to move between the 20 possible depth values.

Prompts

Pipe Depth Options dialog
Pick lower left grid corner [int on]: *pick the profile grid corner*
Pick vertical position for dimensions: *pick a point below the profile grid*



Pipe Depth Summary

Surface Profile: C:\SCAD2005\DATA\SURFACE.PRO
Pipe profile: C:\SCAD2005\DATA\SEWER.PRO

Depth	Manholes	Linear Ft	% of Total
0-2.0	0	0.0	0.0
2.0-4.0	1	34.2	7.9
4.0-6.0	0	280.6	65.3
6.0-8.0	2	115.2	26.8
8.0-10.0	0	0.0	0.0
>10.0	0	0.0	0.0
Total	3	430.0	

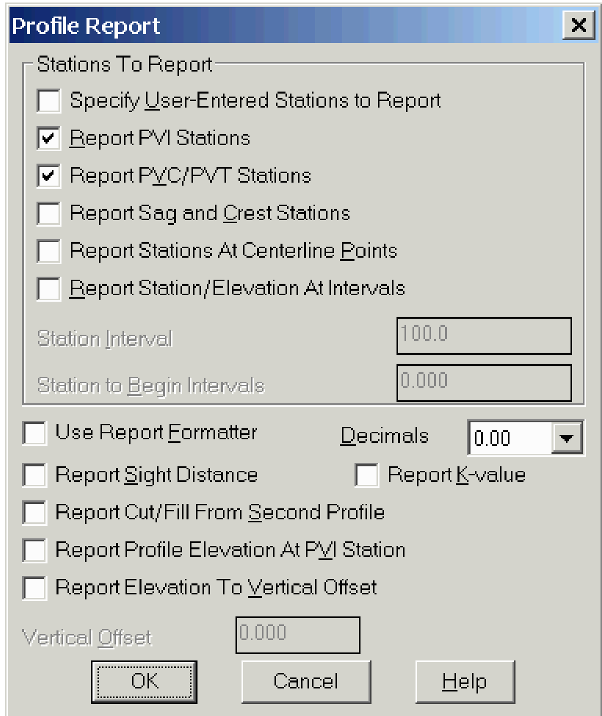
Manhole	Depth
MH#3	6.45
MH#2	6.21
MH#1	3.28
Total:	15.94

Pulldown Menu Location: Profiles
Keyboard Command: pipedeep
Prerequisite: Two profiles, one for the surface and one for the pipe invert elevation
File Name: \lsp\profedit.arx

Profile Report

This command creates a summary report of generic, road, crossing, pipe and sewer profiles using a profile file (.PRO file). The report is generated in the standard report viewer which can print the report, save it to a file or draw it on the screen. The different types of profiles have different report options.

For roadway profiles, Report Sag and Crest Stations will calculate and report sag and crest stations and elevations. Report Stations at Centerline Points will prompt the user for a centerline file (.cl file) and report stations and elevations at horizontal PC and PT points. Report Cut/Fill from Second Profile will compute and report the elevation difference between the subject profile and a second reference profile. Report Station/Elevation at Interval will calculate and report stations at the specified interval in addition to other points. Report Elevation to Vertical Offset creates and additional elevation column in the report. The differential amount for this column is specified by the user in the Vertical Offset window. The Use Report Formatter option runs the report through the report formatter where you can choose which fields to report and the report order as well as output to Excel or databases.



Prompts

Specify a Profile File dialog Choose the .PRO file.
Profile Report dialog Make selections, click OK.
If a vertical offset is entered, a second column of elevations is reported.

Sample Profile Report:

```
 Profile Report
Road Profile
Station Elevation Type VertCurve Distance Slope Desc
0+00.00 88.08          0.00
1+00.00 94.39                             6.45%
2+00.00 100.84                            6.45%
3+00.00 107.29                            6.45%
3+73.78 112.05    PVC           371.48    6.45%
4+00.00 113.68                            6.00%
5+00.00 118.82                            4.27%
6+00.00 122.22                            2.54%
6+23.78 128.18    PI   350.00   250.00    6.45%
7+00.00 121.26                            -6.10%
7+23.78 119.50    PVT           100.00    -8.67%
7+75.71 115.00         0.00     51.93     -8.67%
```

Pulldown Menu Location: Profiles
Keyboard Command: preport
Prerequisite: A .PRO file
File Names: \lsp\proreprt.lsp, \lsp\profedit.arx

Quick Section

This command creates section files in one step. The horizontal alignment for the sections can be defined by using picked points, a centerline file or a polyline. A section alignment (.MXS) file is not required for this routine. 3D screen entities or surface files (.GRD, .FLT, or .TIN) are used to define the vertical alignment.

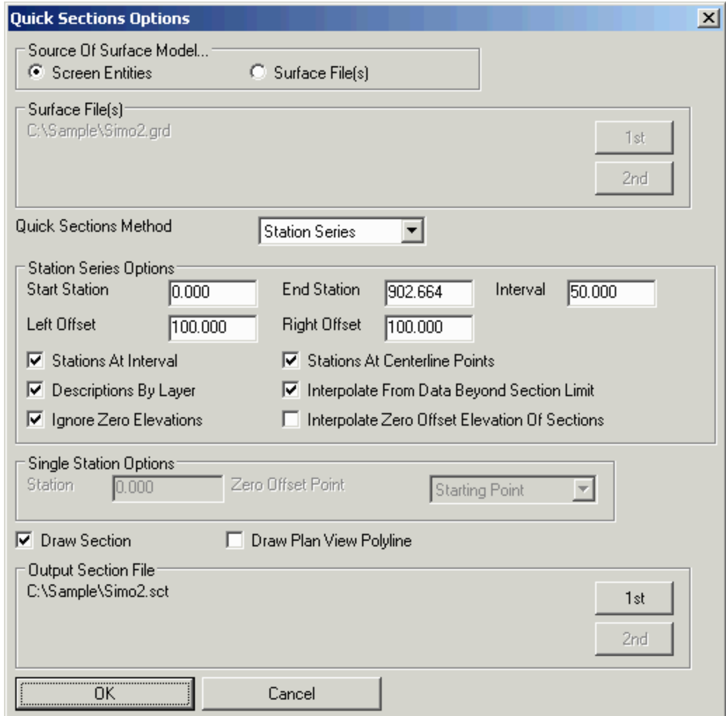
There are two options under Quick Section Methods. The Station Series method creates sections perpendicular from the horizontal alignment at a station interval. In this case, the horizontal alignment represents the centerline. The Single Station method creates one section along the horizontal alignment appends this section to the output section file. In this case, the horizontal alignment represents the alignment of the section.

For the Station Series method, there are settings for the Start Station of the horizontal alignment, the End Station to stop creating sections, the Interval for the stations, and the Left and Right Offsets to define the section width. There are also options to control the section stations to create. The Stations At Interval option will create sections at the specified station interval. The Stations At Centerline Points option will create sections at the special stations of the centerline for the centerline transitions such as PC, PT points

For the Single Station method, the Station value is assigned to this section. The Zero Offset Point chooses between using the starting point of the horizontal alignment as the zero offset or selecting a point along the alignment as the zero offset.

With the Source Of Surface Model set to Surface Files, the program prompts for up to two surface files so that up to two section files can be generated at a time. When the Surface Model is set to Screen Entities, only one section file is created from the screen entities. With Screen Entities, there are a few more options. The Descriptions By Layer option will use the layers of the screen entities as the descriptions for the section points. The Interpolate From Data Beyond Section Limit will check for intersections with the section line and the screen entities beyond the left/right offsets to interpolate the elevations at the left/right offset extents. The Ignore Zero Elevations will filter out screen entities that are at zero elevation. The Interpolate Zero Offset Elevation Of Sections will create a section point at offset zero by interpolating between the nearest section points.

The program requires an output section file to store the results. There is an output option to draw the sections which calls the Draw Section File command. Finally, the option to Draw Plan View Polyline will draw the horizontal alignment as a polyline which is especially useful is the method to define the alignment by picked points was used.



Prompts

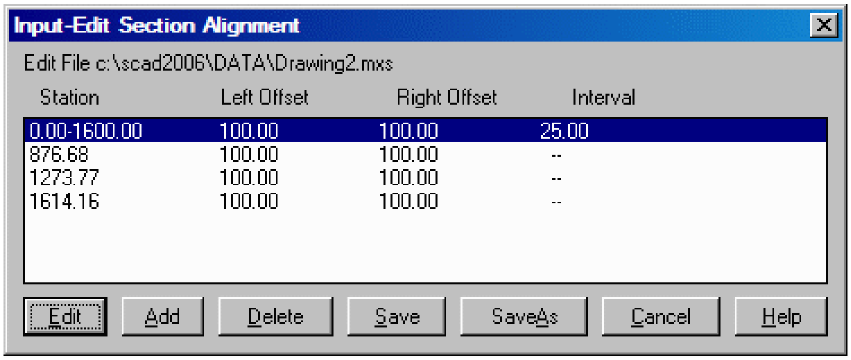
Pick starting point (CL-Centerline,P-Polyline): *select a point*
Pick second point: *select second point*
Pick next point (Enter to end): *press Enter*
Quick Section Options dialog
Choose Source of Surface Model, Screen Entities or Surface File, and make other selections. Click OK.

Keyboard Command: quicksct
Prerequisite: 3D Screen entities or surface files
File Name: \lsp\profedit.arx

Input-Edit Section Alignment

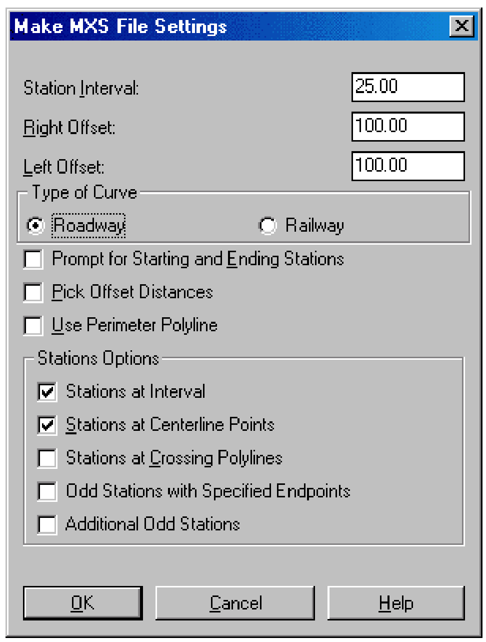
This command will create or append to a section alignment file which is stored as a Multiple Cross Sections (.MXS) file. This file contains the coordinates that define the center and endpoints of section lines and is a requirement of many section commands such as *Sections from Surface Entities* and *Sections to 3D Polyline.* The section alignment defines the stations along a centerline and how far left and right to create cross sections. This routine starts by asking for a new or existing .MXS file name. Then the centerline is specified by either by choosing a centerline file (.CL file) or selecting a polyline that represents the centerline. Next, the program prompts for the starting station of the centerline. If this is a new section alignment, the Make MXS File Settings dialog appears.

The Input-Edit Section Alignment dialog lists all the section stations and offsets in the alignment of an existing .MXS file.



Dialog if using an existing .MXS file

Edit: Allows you to edit the currently highlighted row.
Add: Allows you to add more sections by displaying the Make MXS File Settings dialog (shown below).
Delete: Deletes the currently highlighted row.
Save: Saves the MXS file, exits this dialog and draws the section alignment on the screen using temporary vectors (yellow for left offsets, magenta for right offsets). Any viewport change such as *Redraw* or *Zoom* will cause these vectors to disappear. The draw the section lines with Line entities, use the *Draw Section Alignment* command.
SaveAs: Saves a new MXS file with a user-specified name.



Dialog used for a new section alignment

Station Interval: Enter the station interval for sections.
Right Offset: Enter the width for the sections, right of the centerline. Not available if Pick Offset Distances is checked.
Left Offset: Enter the width for the sections, left of the centerline. Not available if Pick Offset Distances is checked.
Type of Curve: Specify either Roadway or Railroad curve.
Prompt for Starting and Ending Stations: Click or do not click.
Pick Offset Distances: Allows you to specify the offsets by using the distance between two picked points in the drawing.

Use Perimeter Polyline: Allows you to specify a closed polyline that will be used as the limit of the cross sections. The offsets will be contained within this closed polyline.
Station Options: There are five methods for locating the stations:
Stations at Interval: Creates cross sections at the specified interval such as every 25 feet. If the Prompt for Starting and Ending Stations is on, then the program will apply the station interval to the user-specified range of stations. Otherwise the station interval is used along the entire centerline.
Stations at Centerline Points: Creates cross sections at every transition point in the centerline such as the PC, PT, spiral points and end points.
Stations at Crossing Polylines: Allows you to select polylines that cross the centerline and creates cross sections at the intersections of these polylines with the centerline.
Odd Stations with Specified Endpoints: Creates cross sections at stations that are entered or at picked points along the centerline. This option also allows you to pick the left and right offset points which do not have to be perpendicular to the centerline.
Additional Odd Stations: Creates cross sections at the specified stations but the offsets are always perpendicular to the centerline with the user-defined default offset distances.

Prompts

Specify an MXS file dialog Choose new or existing.
Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>: *pick centerline*
Enter Beginning Station of Alignment <0.00>: *press Enter*

Keyboard Command: editmxs
Prerequisite: A polyline centerline or a centerline .CL file
File Name: \lsp\profedit.arx

Sections From Existing Surface

Function

This command will create a section file (.sct) for the existing surface. First, select a .mxs file for section alignment. If you need to create a .mxs file go to Input_Edit Section Alignment. Next, save the file with a new name or override a pervious file and the section file will be created. You can now use the other Section commands to draw, edit, and report from this section.

Prompts

Command: sctgrid2
Reading edges 1393, intersections found 541

Prerequisite: a surface

Keyboard Command: sctgrid2

Sections From Design Surface

Function

This command will create a section file (.sct) for the design surface. First, select a .mxs file for section alignment. If you need to create a .mxs file go to Input_Edit Section Alignment. Next, save the file with a new name or override a pervious file and the section file will be created. You can now use the other Section commands to draw, edit, and report from this section.

Prompts

Command: sctgrid3
Reading edges 1393, intersections found 541

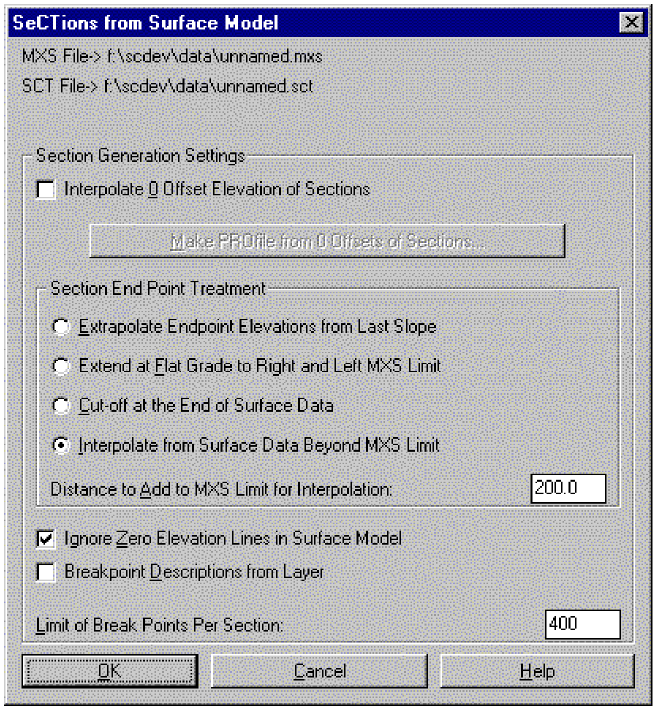
Prerequisite: a surface

Keyboard Command: sctgrid3

Sections from Screen Entities

Function

This command allows you to create cross sections from a surface model. The stations for the sections and the left and right offset distances are defined in the MXS file that must be created before running this routine using Input-Edit Section Alignment. The surface model is defined by lines or polylines with elevation. The polylines with elevation could be a contour drawing file from a photogrammetry firm or can be created from survey points with the *Triangulate & Contour* command. When using Triangulate & Contour, it is useful to use the Draw Triangulation Lines option because the 3D triangulation lines represent all the break lines in the surface which increases the accuarcy of the cross section, as against just using the contours. Barrier lines or 3D polylines can also be used to represent break lines along ridges and valleys. The program samples the selected lines, polylines and 3DFace entities and calculates the intersections of these segments with any of the cross sections. The station, offset and elevation of these intersections make up the data in the section file. This section file (.SCT file extension) can be reviewed or edited with the



Input-Edit SeCTion File command. Also the section file can be plotted with the *Draw Section File* command or used in the by the *Earthworks and Final Contours* command to calculate volumes.

The options for this command are set in the dialog shown below. The Interpolate 0 Offset Elevation of Sections option will add a data point at offset zero for every station with an elevation that is interpolated from existing offsets. The Breakpoint Descriptions from Layer option will store the layer name of the surface entity as the description for the offset-elevation point in the section file. The section end points are the left and right furthest offsets such as left and right 100 feet. When calculating sections based on the intersections with surface entities, there usually, intersection exactly at the end points is not possible. For example, there could be contours at offsets right 87.31 and 105.43 but no intersection exactly at 100. There are four methods for determining the elevation for these end points. (1) The Extrapolate Endpoint Elevation from Last Slope calculates the slope from the last two offset-elevation points and calculates the elevation at the endpoint from this slope. For example, given offsets at 80 with elevation 100 and 90 with elevation 101, the elevation at offset 100 would be 102. (2) Extend at Flat Grade to Right and Left MXS Limit uses the last offset elevation as the end point elevation. For example, if the last offset were 85 with elevation 102, the program would add an offset at 100 with elevation 102. (3) The Cut-off at the End of Surface Data option does not add offsets at the end points. The sections will end at the last offset found in the surface model. (4)Interpolate from Surface Data Beyond MXS Limit looks beyond the offset limit for more intersections with surface entities. Then the endpoint elevation is interpolated between the offsets above and below the endpoint. For example, given offsets at 90 with elevation 101 and at 110 with elevation 103, the endpoint offset at 100 would have elevation 102.

Prompts

MXS File to Process Select the section alignment file

New or Append Choose whether to create a new section file or add to an existing section file

Section File to Create Specify the section file

Select Lines, PLines, and/or 3DFaces that define the surface.

Select objects: *pick the surface entities*

Compiling file c:\scdev\data\simo2.sct

Prerequisite: Construct surface model to be sampled

Sections from Grid or FLT File

Function

This command creates a cross-section file (.SCT file) from a surface model that is defined by a 3D rectangular grid file (.GRD file) or a triangulation file (.TIN). The grid file can be created in the DTM-Contour module with the *Make 3D Grid File* routine. The triangulation file can be created with the Write Triangulation File option in the *Triangulate & Contour* command. This command also requires an .MXS file to define the alignment and stations of the sections. The number of section points created is displayed at the end of the routine.

Prompts

Choose Grid file to process

Choose MXS File to Process

Choose SCT file to write

Found 1410 section points.

Prerequisite: A grid file (.GRD file) or triangulation file (.TIN file) and a cross sections alignment file (.MXS file)

Keyboard Command: profedit

Sections from Polylines

Function

This command allows the user to select a polyline that represents a section in cross section view and writes it to a .SCT file. This can be useful for revising sections or for defining a new one. For example, let's say you have extracted sections from a surface model of the existing ground on a site and have plotted them using the *Draw Section File* command. Now use this command to send the sections to a Section file and compute the earthworks using the *Calculate Sections Volume* command. After selecting the command, the Polyline to Section File dialog appears.

The first time this command is selected the output Section file is set to the same name as the current drawing. Select the Section File Name... button to specify a different name.

The Station Interval edit box allows you to specify the amount that the default station number will be incremented as the station prompt shown below appears.

The Interpolate Zero Offset toggle if on, causes the program to output the elevation of the zero offset to the output .SCT file.

A second section file can be specified to process two sections at a time for each station. This allows you to handle both existing and final grades at once.

After selecting the OK button the prompts below appear.

Prompts

Command:
SCTFPL
Exit/Pick text/<Station <0.0000>>: Press Enter for the default

Exit/Pick text/<Starting elevation of grid <100.0000>>: Pick Text from the screen

[int on] Pick point at starting elevation and zero offset of section ([Enter] for none): Press Enter
Select station 0.0000 section polyline: Pick a Polyline
Renamed original file> C:/TAKEOFF_2004/DEMO2A-OG.SCT as> C:/TAKEOFF_2004/DEMO2A-OG.sck
1 -1766.0303 832.1150
2 -1768.3750 805.5000
3 -1765.7500 780.0000
List continues...
Sta> 0.000 Revised data stored in C:/TAKEOFF_2004/DEMO2A-OG.SCT
Exit/Pick text/<Station <50.0000>>: Press Enter for the default

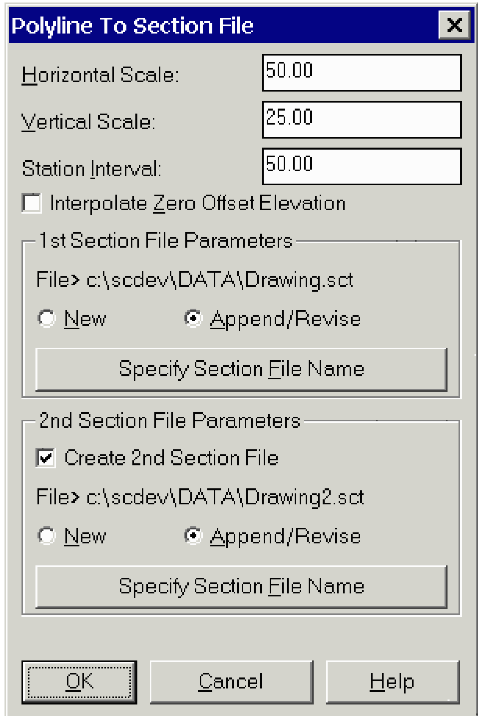
Exit/Pick text/<Starting elevation of grid <100.0000>>: Pick Text from the screen

[int on] Pick point at starting elevation and zero offset of section ([Enter] for none): Press Enter

Select station 50.0000 section polyline: Pick a Polyline

Renamed original file> C:/TAKEOFF_2004/DEMO2A-OG.SCT as> C:/TAKEOFF_2004/DEMO2A-OG.sck
1 1857475.2197 159052.3650
2 1857472.8750 159025.7500
3 1857475.5000 159000.2500
List contiunes...
Sta> 50.000 Revised data stored in C:/TAKEOFF_2004/DEMO2A-OG.SCT
Exit/Pick text/<Station <50.0000>>: Exit
Prerequisite: Plot the section or profile to write to the .SCT file.

Keyboard Command: sctfpl



Sections from Points

Function

This command creates an .SCT file from Carlson points in the drawing. An .MXS file is needed to define the centerline and the stations of the cross sections. The offsets for the cross sections points are derived from the perpendicular distance between the centerline and the TakeOff points. The cross section elevations come directly from the elevations of the points. In order to be included in a cross section, a TakeOff point must be within the offset tolerance distance of the cross section line.

Prompts

Choose MXS File to Process

Choose SCT file to write

Enter the maximum offset tolerance <1.0: *Press Enter*

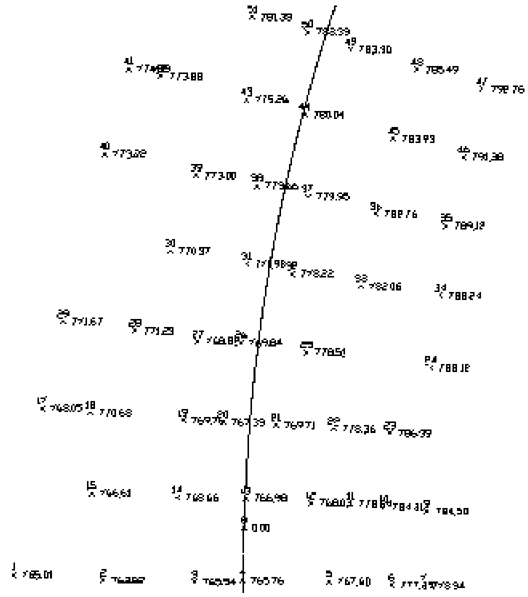
Ignore Zero Elevations (<Yes>/No)? *Press Enter*. This option will filter out all Carlson points that have a zero elevation.

Select the Carlson points along the sections.

Select objects: *pick the Carlson point inserts*

Prerequisite: Carlson points and an MXS file

Keyboard Command: sctpts



TakeOff points for use in creating Section file

Import Sections

Function

This command converts Columnar Text, Agtek, Ceal, GeoPak, IGRDS, Moss, RoadCalc, SMI, and Softdesk files into Carlson TakeOff section (.section) files.

Prerequisite: a Columnar Text, Agtek, Ceal, GeoPak, IGRDS, Moss, RoadCalc, SMI, or Softdesk file

Sections to 3D Polylines

This command creates 3D polylines from a section (.SCT) file. Besides the section file, a centerline polyline, centerline file or section alignment (.MXS) file must be specified to define the plan view location of the 3D polylines. The elevations for the 3D polylines come from the section file. These 3D polylines can then be used by other Carlson routines to create surface models.

Typically, the 3D polylines are drawn as cross-sections perpendicular to the centerline at each station. When using a polyline centerline instead of the .MXS file, there is an option to draw by connecting similar descriptions to make 3D polylines parallel to the centerline. For example, if the section file has descriptions for each section point then you can draw 3D polylines for EP, SHD, TIE, etc.

Prompts

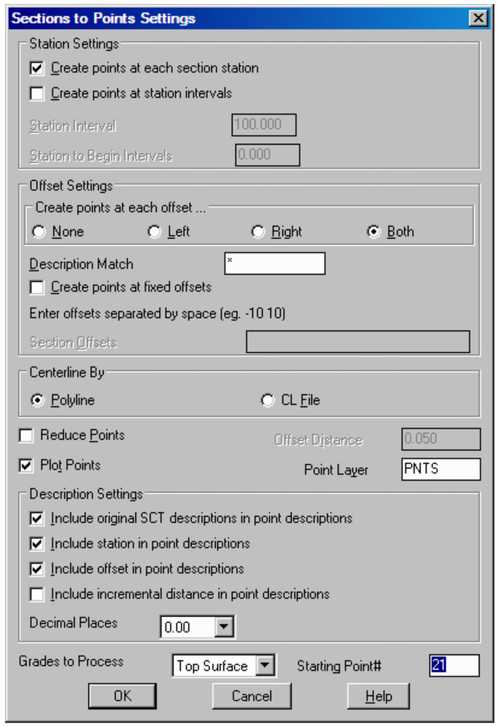
Layer Name for 3D Polylines <3DXSEC>: *press Enter*

Align sections by MXS file, centerline file or polyline [MXS/Centerline/<Polyline>]? *press Enter*
Choose Section File to Process Select the .sct file
Range of stations: 1.14 to 1605.25
Enter the starting station to process <1.14>: *press Enter*
Enter the ending station to process <1605.25>: *press Enter*
Draw sections or offset polylines by description [<Section>/Offset]? *press Enter*
Type of centerline [<ROadway>/RAilroad]? *press Enter.* This option chooses between roadway and railroad methods for stationing along curves.
Select centerline polyline: *pick the polyline*
Enter the centerline starting station <0.0>: *press Enter*
Draw perimeter of sections [Yes/<No>]? *Y* This option will connect all the left most offsets and right most offsets together with a 3D polyline.
Use reference profile to interpolate between sections [<Yes>/No]?*N for no.* This option will prompt for a profile to use for interpolating elevations along the 3D polylines between the section stations. This improves the accuracy when the profile goes through vertical curves. Without the profile, the 3D polyline elevations will be straightline interpolated between the sections.
Draw all template ids or specific ids and offsets [All/<Specific>]? *press Enter for Specific*
Enter Offset or Description to draw: *EP*

Keyboard Command: scto3dp
Prerequisite: A section (.SCT) file
File Name: \lsp\scto3dp.lsp, \lsp\plinsct.arx, \lsp\profedit.arx

Sections to Points

This command creates Carlson points using a section (.SCT) file to define the point elevations. The x,y position of the points are calculated based on the station and offset along a centerline polyline. These points are stored in the current coordinate (.CRD) file and can also be plotted in the drawing. Points can be created at each station in the section file or at a set station interval. The range of stations to process can also be set. The Description Match field can be used to filter the offsets and only create points with matching descriptions (e.g. only "EOP" offsets). The Create points at fixed offsets option can be used to make points at user-specified offset distances. The program will interpolate the elevations for these points by interpolating from the neighboring offsets. The is both a Centerline by Polyline or by CL File option. The CL File option will prompt for an existing centerline (.CL) file. The Reduce Points option will skip creating points for the same offset between stations if the x,y position and elevation change is less than the offset tolerance. Essentially, when a series of offsets are on a straight line (no vertical and no horizontal curve) then only the starting and ending points are needed and all the intermediate points can be skipped. For example, the Reduce Points routine will look at the left side EOP offset points at stations 1+00, 1+05 and 1+10 and if these three points make a straight line then the point for station 1+05 can be reduced. The Offset Distance is the tolerance that Reduce Points using for testing whether the middle point (offset point at station 1+05) can be reduced. The distance for the middle point is calculated as the perpendicular distance from the middle point to the line between the two end points. Both the horizontal and vertical distances are checked.



Prompts

Sections to Points Settings dialog
Coordinate File to Process Choose a .CRD or other coordinate file to add the points to. This prompt only occurs if no coordinate file is current.
Choose SCT file to read *pick the cross section file*
Range of stations: 3.34 to 750.00
Enter the starting station to process <3.34>: *press Enter*
Enter the ending station to process <750.00>: *press Enter*
Select centerline polyline: *pick the polyline that defines the stations*
Type of centerline [<ROadway>/RAilroad]? *RO*
Enter the centerline starting station <0.0>: *press Enter*
Created 65 points.

Keyboard Command: sctopt
Prerequisite: A .sct file and polyline centerline
File Name: \lsp\plinsct.lsp

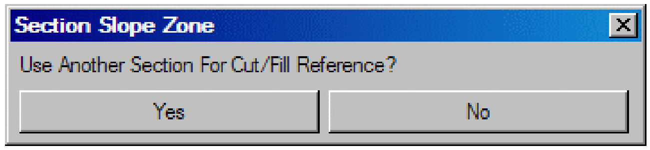
Slope Zone Section Analysis

This command reports the cut/fill areas and volumes within given ranges of slopes. There is an option to use another section for cut/fill reference.

Prompts

Select Section to Process ***Select .SCT file***

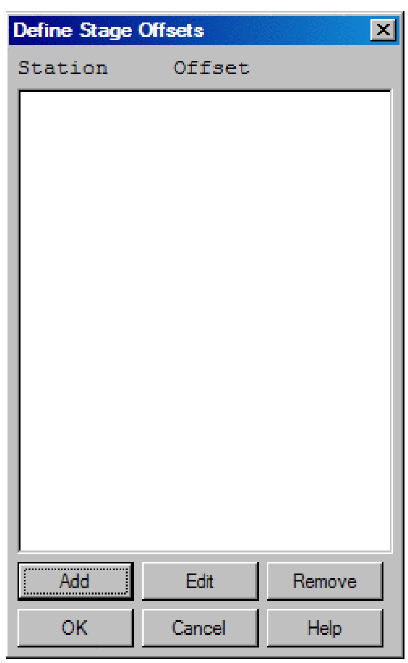
Select Slope Zone dialog ***Select No***
Report slope or horizontal area [<Horizontal>/Slope]? *s*
Slope format [<Percent>/Ratio]? *Enter*
Greatest slope % of zone 1: ***3***
Greatest slope % of zone 2: *Enter*
Starting station to process <0.000>:
Ending station to process <0.000>: 1000
The Standard Report Viewer creates a report called Section Slope Zone Analysis Report.



Keyboard Command: sctzone
Prerequisite: .SCT file
File Name: \lsp\regrade.arx

Highway Section Staging

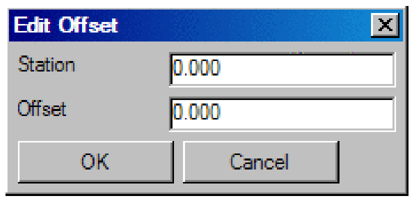
This command applies to the situation of designing a partly completed road or regrade. Using an existing and a final grade section file, the program will create four new sections files for the finished existing sections, finished final sections, remaining existing sections, and remaining final sections. The source existing and final section files should have matching stations. There is an option to process a range of the possible stations from the section files. The complete part of the road can be either on the left or right side. The pivot point is a cross section offset where the completed part ends. From this point, the final grade will connect to the existing grade by a line at the specified slope.

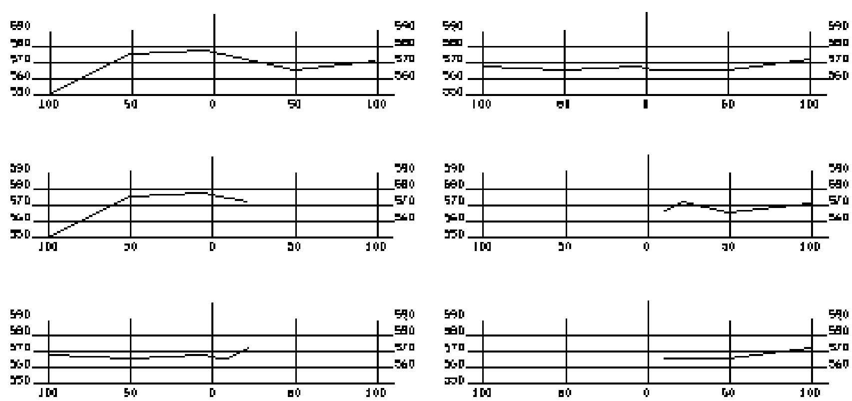


Prompts

Select Existing Sections File Choose the cross sections file.
Select Final Sections File Choose the cross sections file.
Enter slope as percent grade or slope ratio (Percent/<Ratio>)? *press Enter*
Enter the fill slope ratio <2.0>: *press Enter*
Enter the cut slope ratio <2.0>: *press Enter*
Place road on left or right (<Left>/Right)? *press Enter*
Range of stations: 50.0 to 100.0
Enter the starting station to process <50.0>: *press Enter*
Enter the ending station to process <100.0>: *press Enter*
Apply same pivot offset to all stations (Yes/<No>)? *Y*
Enter the pivot offset (enter left offsets as negative) <0.0>: *5.0*
SCT File dialogs Enter new .SCT file names for 1) existing road .SCT file, 2) final road .SCT file, 3) remaining existing .SCT file and 4) remaining final .SCT file.

As mentioned above, these are the four new sections files for the finished existing sections, finished final sections, remaining existing sections, and remaining final sections that the routine creates.





Keyboard Command: sctstage
Prerequisite: Existing and final grade section files (.SCT)
File Names: \lsp\sctstage.lsp, \lsp\profedit.arx

Input-Edit Section File

Function

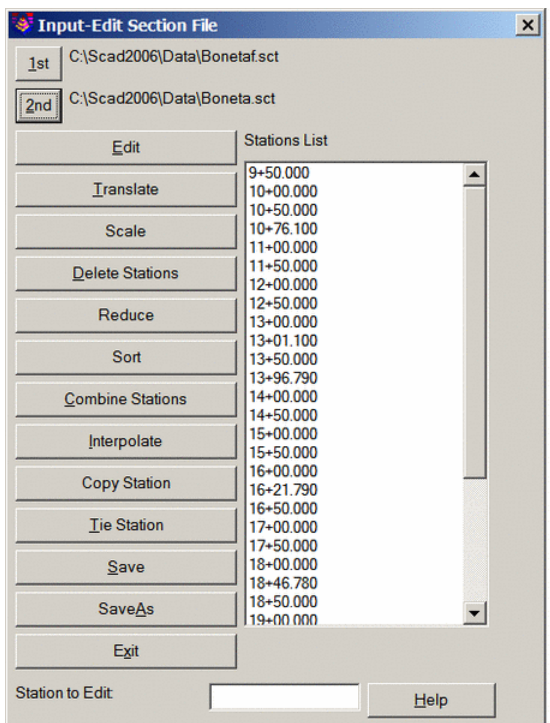
This program can be used to enter or edit data stored in a section file (.SCT file), including a real-time graphic window in the Edit mode. The section data consists of stations, offsets, elevations and descriptions. This command also has utilities for translating the offsets and elevations, deleting stations from the file, intersecting the outslopes of one section file with another, combining multiple occurrences of the same station and sorting the stations, offsets and elevations. While editing the section file, a second section file can be used as reference. To choose this file, pick

the 2nd button. For example, when editing the proposed section file, you can reference and view the ground section file as the 2nd file.

The program begins by prompting for a New or Existing section .SCT file to process. The Section file to process dialog appears, allowing you to specify the file that you want to operate on. Use the New option to create a new file. Use the Existing option to edit the offsets and elevations for station/sections that you have already created, or append new stations to a file. The program defaults to a section file with the same name as the drawing or a name that you specified using another section command. You also can choose a 2nd existing .SCT file to reference. After specifying the file name(s), the program displays any stations currently in the file, in the Stations List of the Input-Edit Section File dialog box.

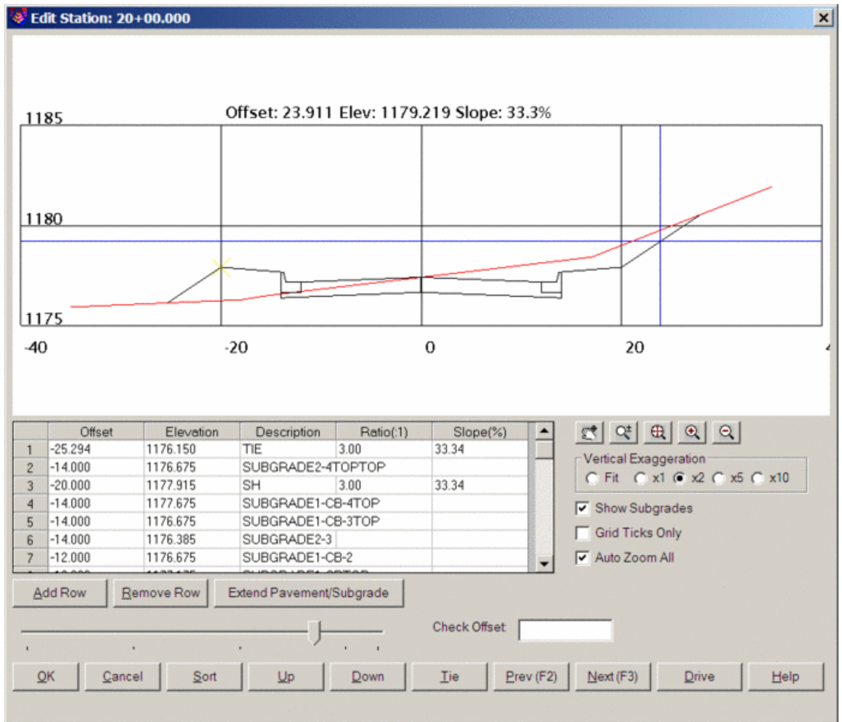
Alternately, when sections are drawn in the drawing, you can double-click on a section polyline to launch Input-Edit Section File for the .SCT file associated with the section polyline.

If you specified a new file, the Stations List box will be blank. To edit and display the offset and elevation data at a station, you can double click on the station in the Stations List box, or input the station in the Station to Edit edit box at the bottom of the dialog. To add a station to a new file or existing file, you must enter the station in the Station to Edit edit box. Stations will present in accordance with the Section-Profile settings in Configure under the Settings pulldown menu (eg. 10+00, 1+000, 1000).

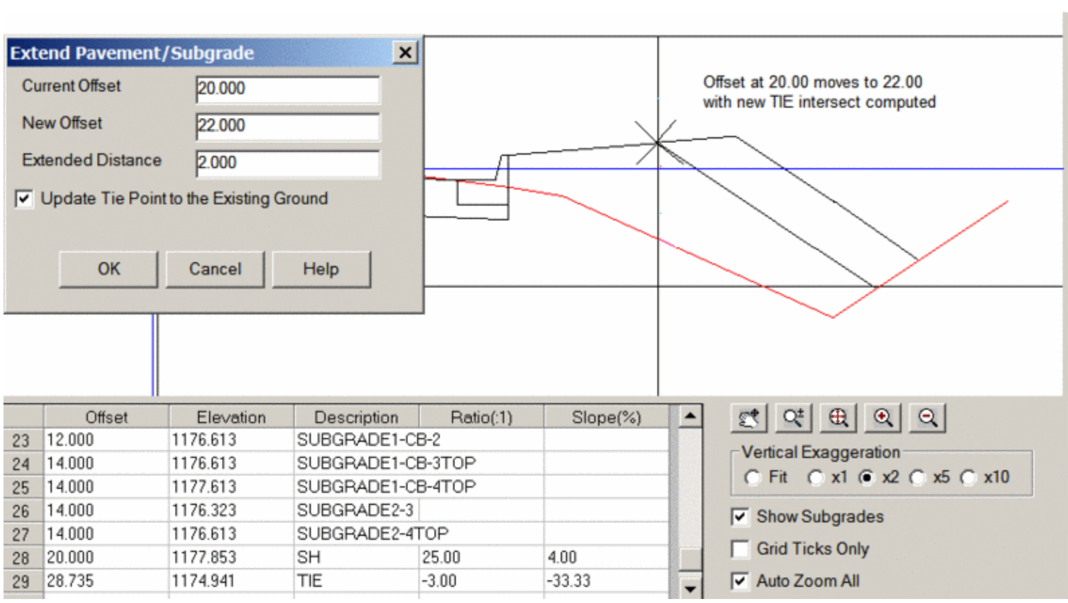


Edit: Opens the Edit Station dialog which shows a graphic of the section on top, a list of the offset-elevation points in the middle, and the function buttons on the bottom. To add an offset point, type in the offset, elevation and optional description in the spreadsheet. Left offsets are entered as negative numbers. You can enter the slope or ratio from the last point and the program will calculate the elevation. To edit an offset point, highlight the point from the list and then edit the values in the Offset, Elev and Desc columns. The highlighted point will be marked by an X in the graphic screen. The Sort button will sort the list of offsets from lowest to highest, left to right. It is recommended that you Sort offsets before doing the Tie command, so that the left-most and right-most offsets appear first and last in the offset list. The Up button will move the highlighted offset point up in the list. Likewise the Down button moves the highlighted offset point down in the list. Prev (F2) and Next (F3) buttons move through the stations and

allow you to review and edit stations in forward or reverse order. The scroll bar can also be used to quickly move through stations and then zero in with Prev (F2) or Next (F3).

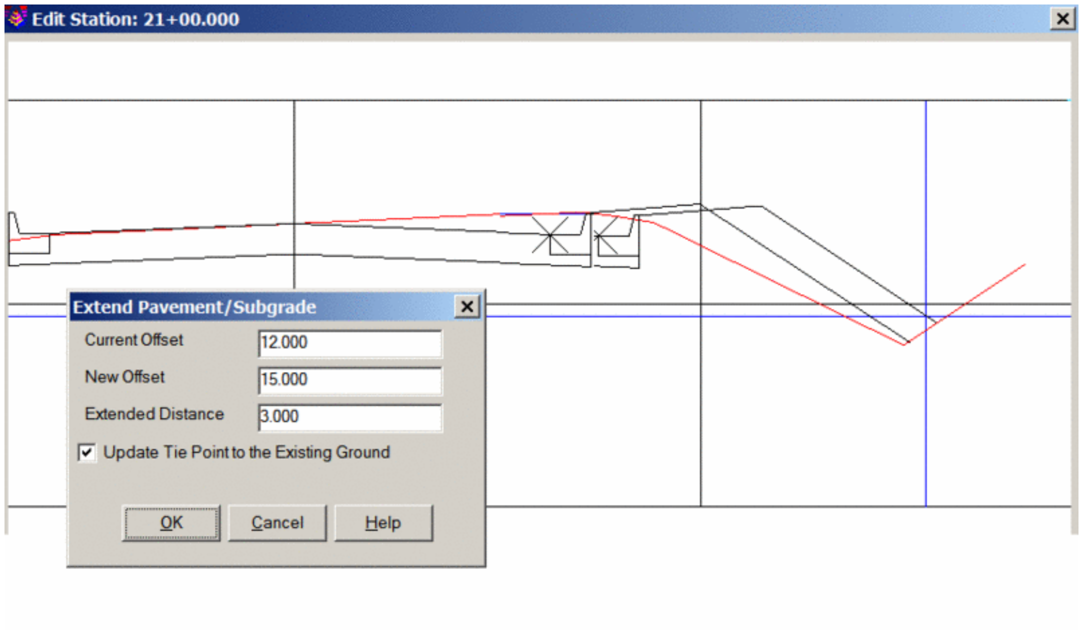


The Add Row button inserts an offset line above the currently highlighted row. The Remove Row button erases the highlighted offset and elevation from the list. After inputting or editing press the OK button to return to the Stations List dialog and keep any changes you have made. Select the Cancel button if you want to cancel changes made to the current station. Extend Pavement/Subgrade will allow you move a surface point and shift, in parallel, the associated subgrades and tie points. One application, shown below, is to extend a shoulder point and re-computer the TIE point, all in one clean operation:



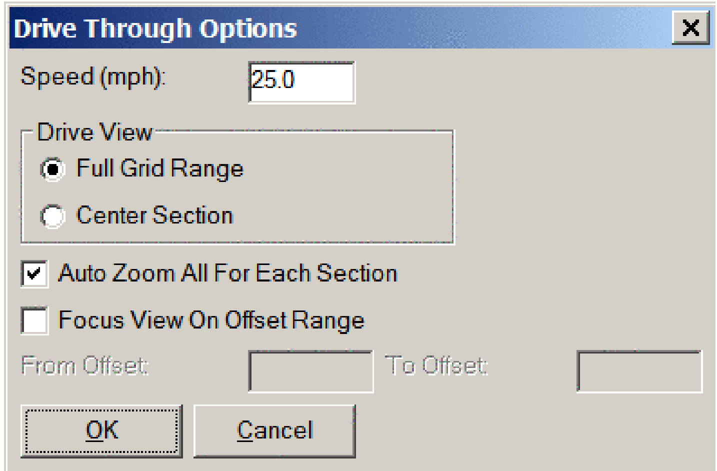
Another application of Extend Pavement/Subgrade is to move the curb position and all associated subgrades. The

"inside" curb point is at 12.00 units from centerline. If the pavement is extended from 12 to 15 at this station, use of this feature will extend the subgrades, maintain all slopes and re-compute the TIE point, as shown below:



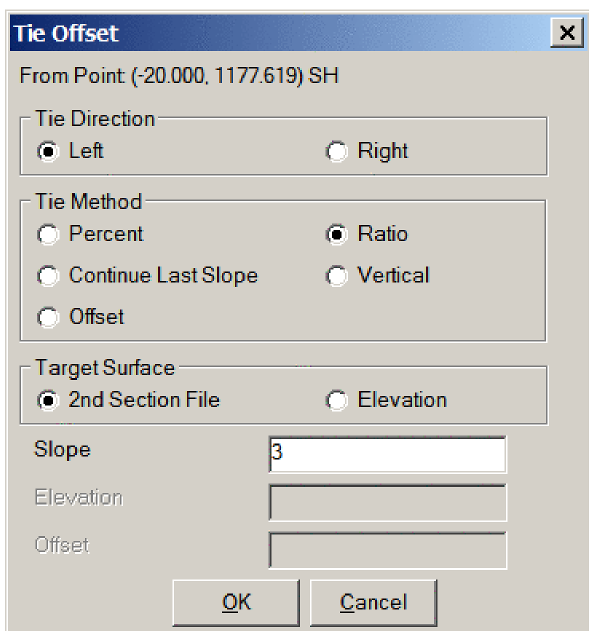
A real-time report of offset-elevation-slope now displays in the top of the graphic as you move the cursor across the section in the graphic window. The screen defaults to zoom mode where holding down the right-mouse button zooms in and out. You can also switch to pan mode. There are buttons for zoom extents, zoom in and zoom out. If your mouse has a scroll button, you can hold it down to pan and scroll it to zoom in and out. You can also set the Vertical Exaggeration ranging from 1X to 10X and including "Fit". Show subgrades has the ability to tie a subgrade into the surface. Grid Ticks Only just shows the left and bottom axis lines of the grid with grid tick marks along the axes. With Auto Zoom All turned off, you can hold the same view position as you click Next and Previous and move through the list of stations. The Check Offset field calculates an elevation based on an entered offset.

Drive (Edit Station): This function scrolls through the sections at the rate of speed specified by the user in the Speed window. The Drive View options determine whether the sections are displayed using the full width of the graphic window or centered in the window. The combination of Full Grid Range and Auto Zoom All allows the sections to rise and fall with the centerline elevations, as if you were driving an actual road. With Auto Zoom All off, and Full Grid Range on, the grid itself moves up and down at the current position of the first section, as you drive. Focus View On Offset Range allows the user to set the left and right viewing limits of the sections. Section data beyond the specified limits is not displayed.



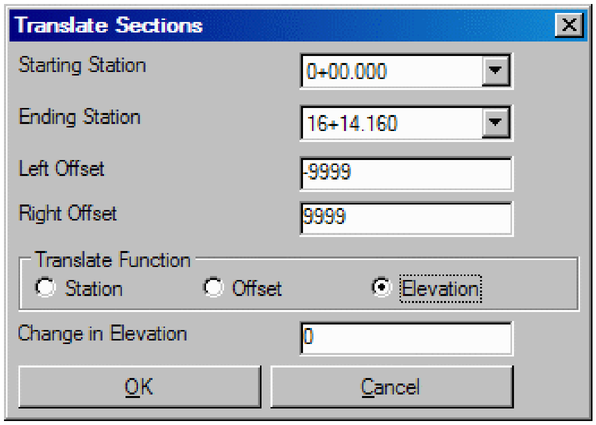
Elevation Field (Edit Station): Equations (+, -, ∗, /) can be entered to calculate or adjust an elevation. For instance, to subtract 1.25' from elevation 1926.18, simply enter 1926.18-1.25 and press enter. The new elevation will be calculated and displayed in the viewer window.

Tie (Edit Station): The Tie button allows you to tie the left and right surface points of the 1st section file into the 2nd section file. It is used for classic outslope intersects from final grade to existing grade. The dialog layout includes an option to tie the section to a specified elevation, in addition to a surface (second section file). A left or right tie direction can also be selected. If a point has been tied in from SH for shoulder at offset -20 at 3:1, a new offset with the description "TIE" is created. If you try another outslope such as 4:1 from the same SH shoulder point, a new "TIE" point is created and the old TIE point is removed automatically.

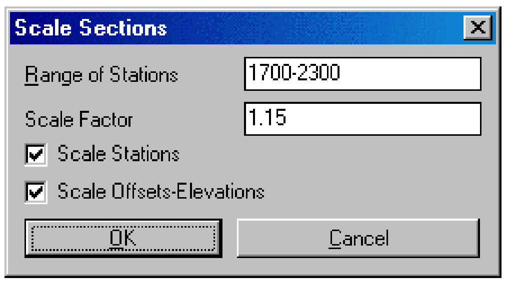


Translate: Allows you to add or subtract a distance from the offsets to adjust or shift the centerline. You can also adjust the elevations up or down. When using this option, you can choose the range of stations to operate on (starting and ending stations) and the values to adjust the offsets and elevations. If, for example, you want to shift the centerline, but not the elevations, enter the plus or minus amount you want to translate, and when prompted for

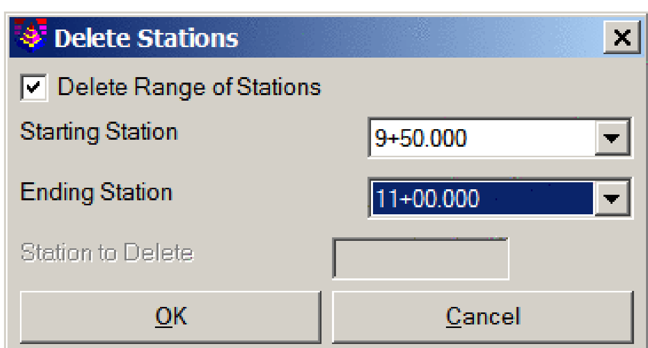
the elevation enter zero.



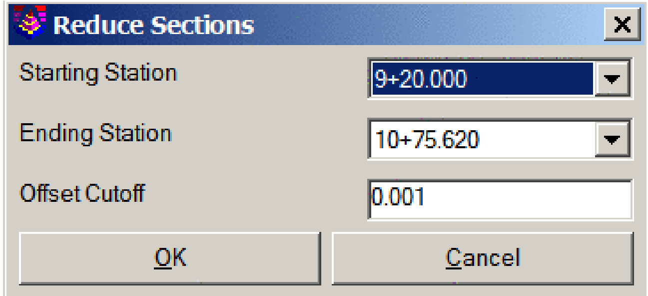
Scale: Allows you to scale the station, offsets and/or elevations by the specified scale factor. This function can be used to convert between English and metric units.



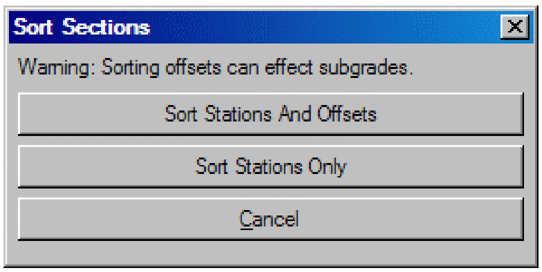
Delete Stations: Allows you to remove a station or range of stations from the in memory Stations List. Click "Delete Range of Stations" on to delete a range, a click it off to delete an individual station. Since the station editor data is stored in virtual memory, if you accidentally delete a range, Quit the editor with out saving the stations to disk. Then recall the original file.



Reduce: Allows you to remove offsets from a range of stations by removing vertices in the offsets that are virtually in a straight line. Using an offset cutoff, meaning no offset and elevation moves more than the entered amount (eg. 0.01), excessive numbers of vertices can be eliminated. The command is similar to Reduce Vertices when applied to the plan view.

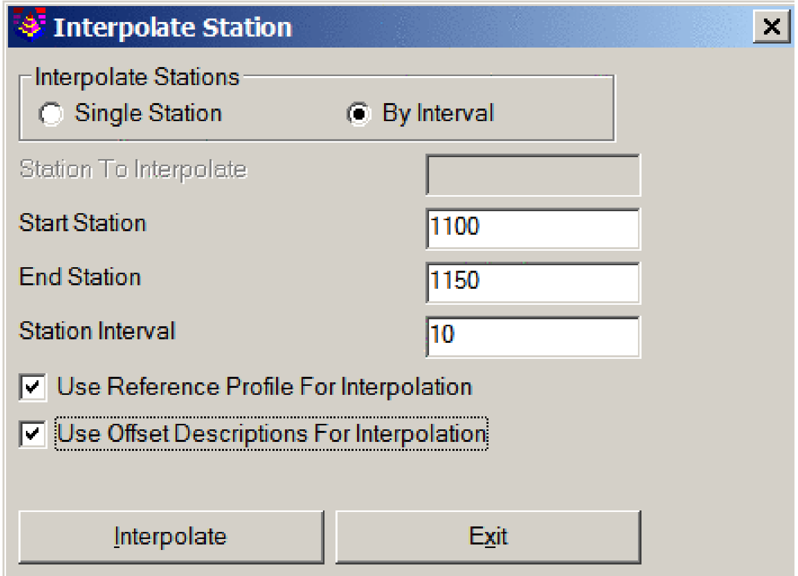


Sort: Allows you to sort the station numbers into ascending order, and sort the offsets and elevations in the individual station records (offsets are sorted from left to right). When sections are derived from the *Sections from Surface Entities* command they are already sorted, but when sections are digitized or input manually they occur in the order that you digitized them. So, for proper plotting and earthworks, you may want to run the Sort option before processing.



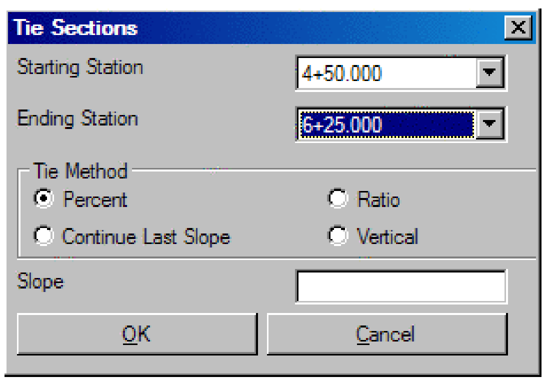
Combine Stations: Used to bring together in one record slot multiple occurrences of the same station number. This can occur when using the Digitize Sections (XSec) command and the section that you are digitizing has match/break lines which forces you to digitize the station in two or more parts.

Interpolate: Allows you to add or overwrite a station between two stations or projecting forward from two stations. You can choose to interpolate a single station or an interval of stations. Specify the two known stations in the Start Station and End Station edit boxes, as well as the interval if using the interval method. The program will do straight line, mathematical interpolations, adding offsets to the interpolated stations to match the totality of offsets in the starting and ending stations. However, if the offsets have descriptions, you can choose to interpolate by description and the program will interpolate by description (eg. EP at 12 on Station 1100 and EP at 15 at station 1150 would lead to EP at 12.6 at 1110). There is also an option to reference a profile, so if station 1100 and 1150 were on either side of a high point at 1125, the interpolated offsets would respect the profile as well as the starting and ending station. Use of this command is often critical to creating accurate digital terrain models of sites for machine control. Select the OK button to execute the function with the current settings or select the Cancel button to abort the process.



Copy Station: Allows you to copy a station that already exists to a new or existing station number. Choose the existing From Station using the edit pulldown box, then enter the new station number in the To Station edit box. Select the OK button to execute the function with the current settings, or select the Cancel button to abort the process.

Tie Station: Allows you to tie the outslopes into the reference second section file. This routine first brings up a dialog to specify the range of stations to process. It includes a line to set the slope to tie with. The program will start from the left most offset and use this slope to find the intersection with the reference section file. Then the intersection from the right most offset is calculated with this slope. These intersection points are the tie points. The slope can be defined by percent, ratio, continue the last slope, and vertical.



Save: Saves the currently loaded section file.
SaveAs: Allows you to save the currently loaded section file as a different file.
Exit: Allows you to exit from the section editor and return to the drawing editor. The program will warn you to save to a file if you have made changes.

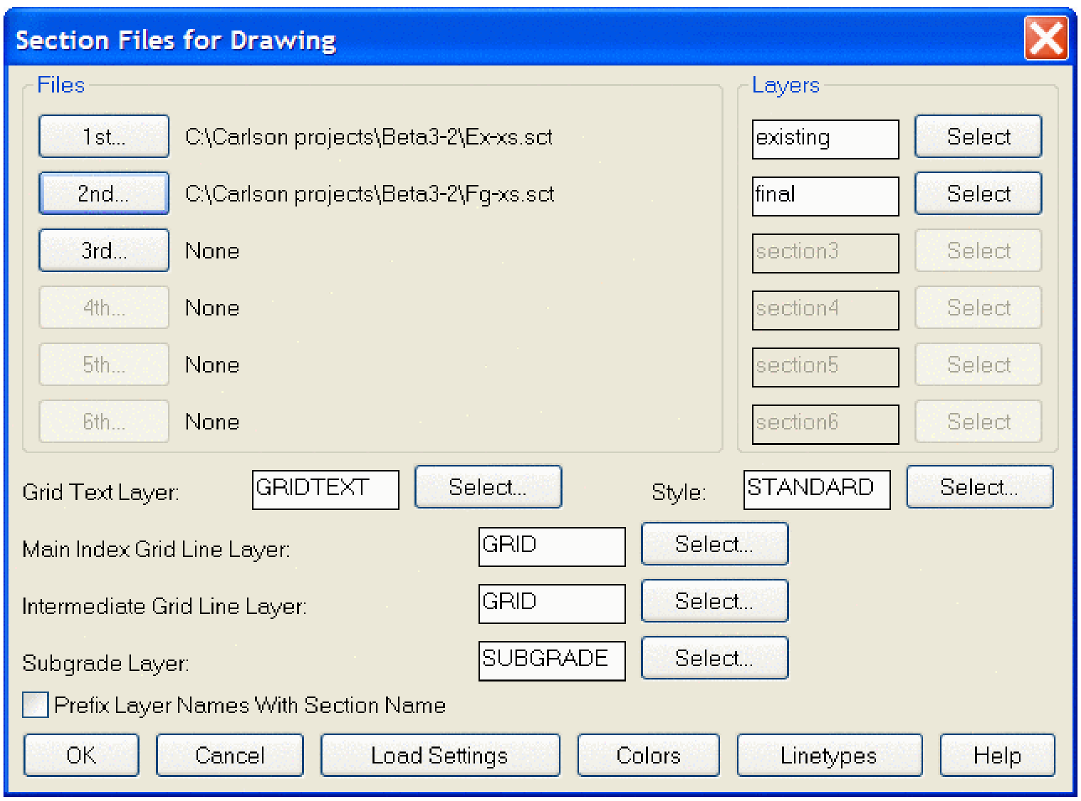
Keyboard Command: scted
Prerequisite: None
File Name: \lsp\regrade.arx

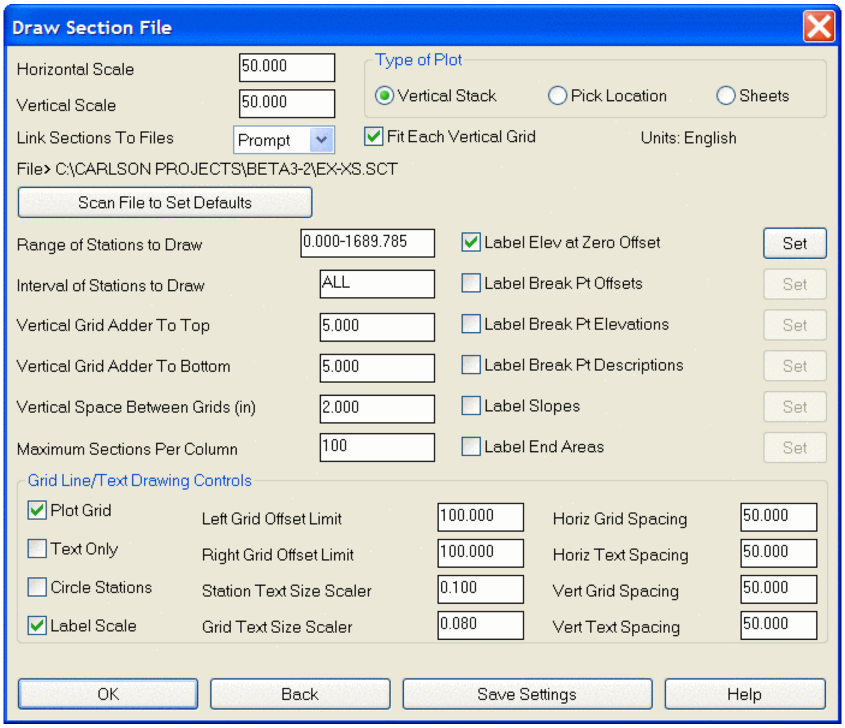
Draw Section File

This command will plot the section data from up to six section (.SCT) files at once. The section file can be created by several methods including *Input-Edit Section File*, *Sections from Surface Entities*, *Digitize Sections* or *Process Road Design* command. A range of sections can be plotted in a vertical stack, on section sheets, or by selecting a point that corresponds to the grid bottom elevation.

When drawing sheets format in metric mode, be sure to set metric On (clicked) in the Drawing Setup command. Then in the Sheet Parameters dialog, set the Block Name to SCTSHT2 and set your metric sizes.

In the initial dialog, specify up to six section (.SCT) files to plot, the layer for each, and the layer names and text styles for the overall grid text, grid lines, and subgrade. There is also a toggle to Prefix Layer Names with Section Name, so that all layers created for the sections begin with the section name. At the bottom of the dialog is a button to Load Settings, loading a set of previously saved settings, and buttons to set the Colors and Linetypes for the section components.



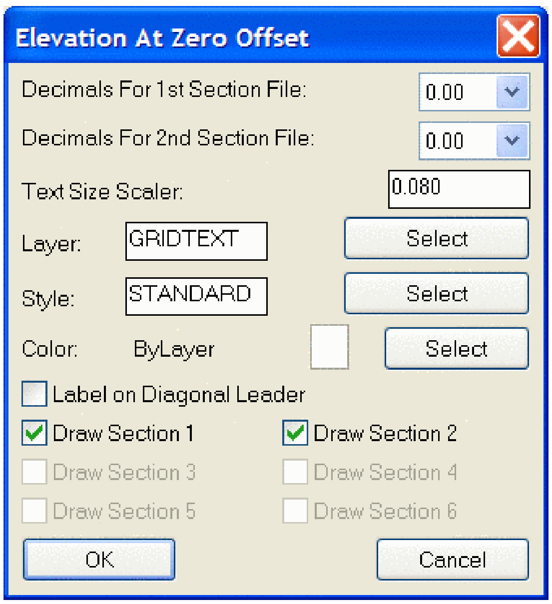


The second dialog box presents the next level of settings for the generation of sections.

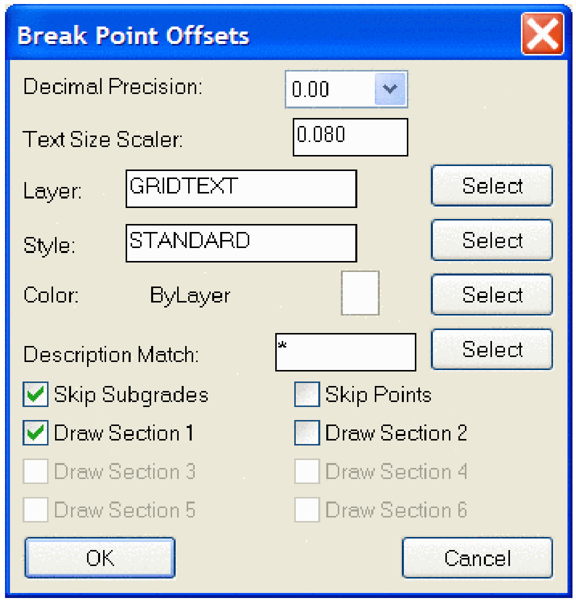
Horizontal Scale: Specify the horizontal scale.
Vertical Scale: Specify the vertical scale.
Link Sections To Files: This setting controls the linkage of the plotted sections to the actual section file(s) (.SCT), determining how changes to the file affect the plotted sections. If set to Off, there is no linkage, Prompt will ask whether to update the plotted sections when the file changes, and Auto will automatically update the plotted sections when the file changes.
Type of Plot: Specify how the sections will be plotted, either as a Vertical Stack, Pick Location, selecting the datum point of each section, or Sheets, which will plot the sections on a block section sheet.
Fit Each Vertical Grid: When checked, the grid bottom elevation and grid height are set automatically, and you may specify values to add to the top and bottom of each grid. See Vertical Grid Adder to Top and Vertical Grid Adder to Bottom. When not checked, the Vertical Grid Adder to Top and Vertical Grid Adder to Bottom options change to Grid Bottom Elevation and Grid Vertical Height.
Scan File to Set Defaults: This button allows the program to set the minimum and maximum parameters. If you choose this option the program will automatically set the range of stations, vertical spacing distance, right and left grid distances and starting/datum elevation. This option writes a file called "sectsort.tmp" that is read and used to set the defaults the next time you use the program. Therefore, if you are selecting a different .SCT file to plot you should use this option to update the .TMP file.
Range of Stations to Draw: Specify the range of stations from the file which will be drawn.
Interval of Stations to Draw: Specify the interval of stations to draw. For example, perhaps you sampled every 25 feet with the *Sections from Surface Model* command for more accurate quantities but only want to plot 50 foot stations. ALL is the default value for this field.
Vertical Grid Adder to Top: Specify the distance that will be added to the highest elevation of the section for the sheets and pick location options. Only available when Fit Each Vertical Grid is checked ON.
Vertical Grid Adder to Bottom: Specify the distance that will be subtracted from the lowest elevation of the section for the sheets and pick location options. Only available when Fit Each Vertical Grid is checked ON.
Grid Bottom Elevation: Specify actual bottom elevation for each section grid. Only available when Fit Each

Vertical Grid is checked OFF.
Vertical Grid Height: Specify actual grid height for each section grid. Only available when Fit Each Vertical Grid is checked OFF.
Vertical Space Between Grids: Specify the distance the sections are stacked above the last one plotted when drawing multiple sections.
Maximum Sections Per Column: Sets the maximum number of sections allowed per column.

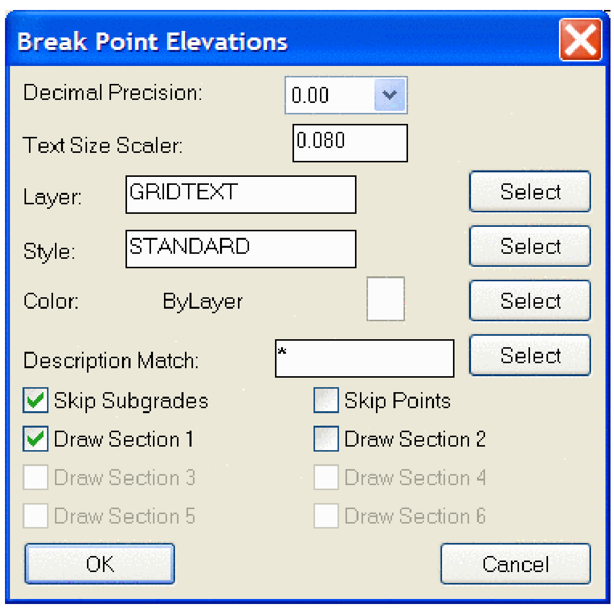
Label Elevation at Zero Offset: Will label the section elevation at offset zero. The label is drawn on the section grid just above the section line. Press the Set button to the right of this toggle to set the display precision, text size scaler and layer for these labels. There is also an option to draw the elevation on a 45 degree diagonal, otherwise the elevation label is draw vertically.



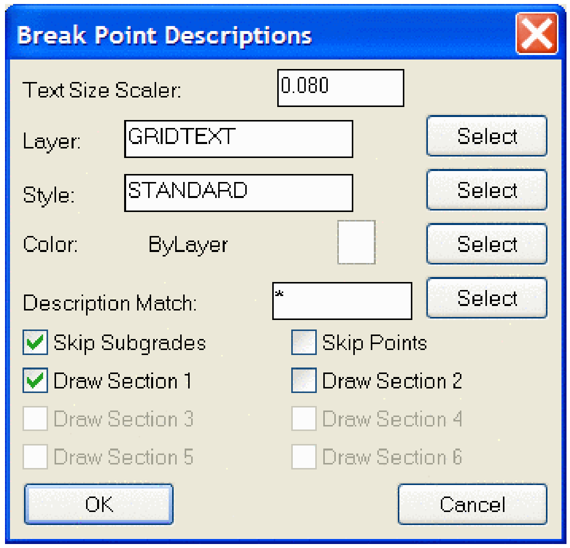
Label Break Pt Offsets: Will label these values along the section line above each point in the section. Press the Set button to the right of this toggle to set the display precision, text size scaler and layer for these labels.



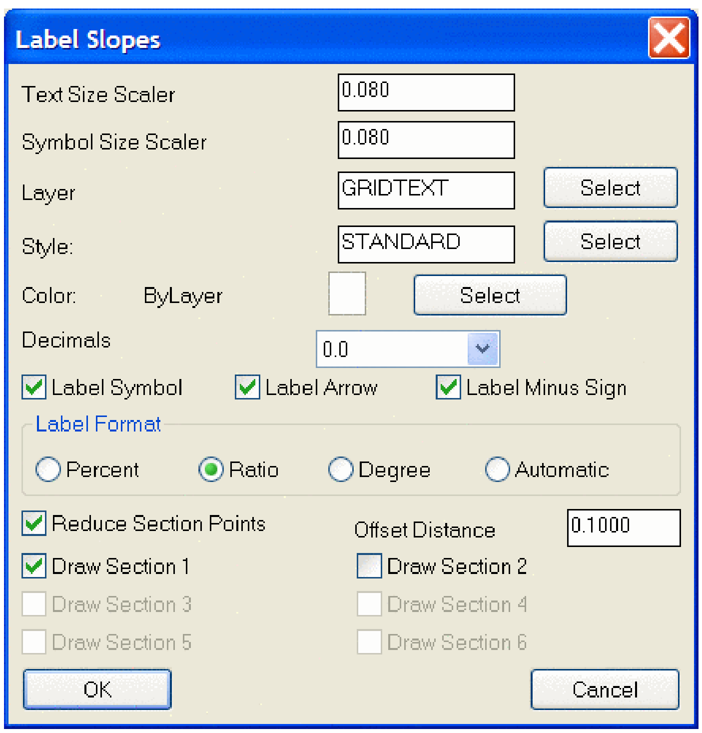
Label Break Pt Elevations: Will label these values along the section line above each point in the section. Press the Set button to the right of this toggle to set the display precision, text size scaler and layer for these labels.



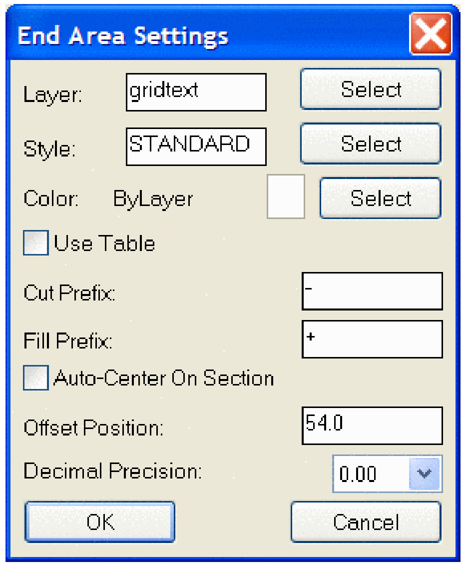
Label Break Pt Descriptions: Will label these values along the section line above each point in the section. Press the Set button to the right of this toggle to set the text size scaler, layer, and description match for these labels.



Label Slopes: Will draw in the slopes.



Label End Areas: Will label cut and fill quantities on each section.
Use Table: Puts end areas cut/fill values in table.



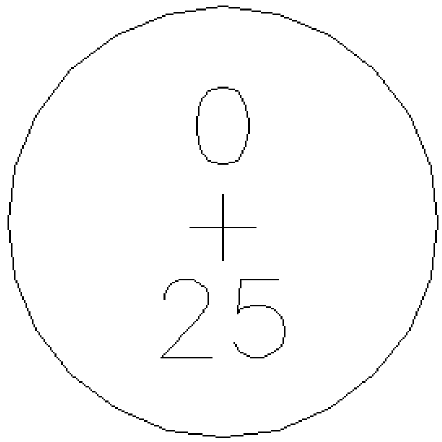
Grid Line/Text Drawing Controls

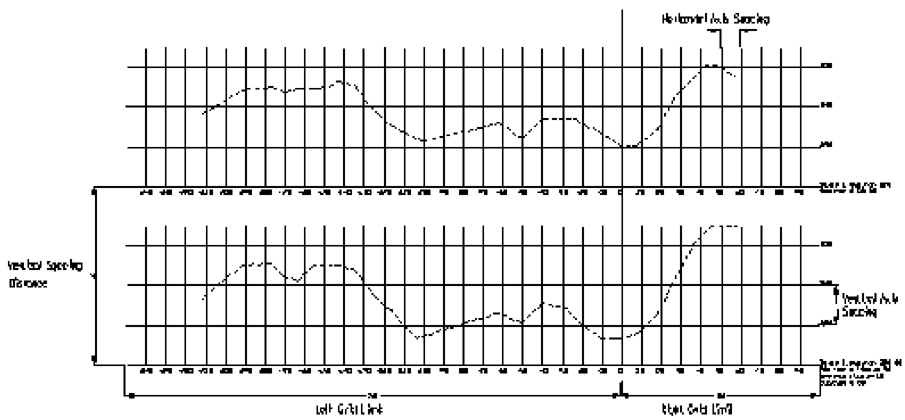
Plot Grid: Uncheck this toggle if you do not want the grid to plot.
Text Only: Check this toggle if you only want to plot the cross section polyline and the grid text. This can be useful for plotting on a section sheet that has pre-plotted grid lines and you want to plot only the section and text.
Circle Stations: Will draw the station number with a circle around it on the left and right sides of the section grid.
Label Scale: Will label the horizontal and vertical scale on each section.
Left Grid Offset Limit: Specify the length the grid lines are plotted to the left from the centerline or zero offset.
Right Grid Offset Limit: Specify the length the grid lines are plotted to the right from the centerline or zero offset.
Station Text Size Scaler: Specify the text size scaler for the station text. This value is multiplied by the horizontal scale to obtain the final text height. For example, if you set Station Text Size to 0.10 and the horizontal scale is 100.0, then the text height will be (0.10 X 100) or 10.0.
Grid Text Size Scaler: Specify the text size scaler for the axis text. This value is multiplied by the horizontal scale to obtain the final text height. For example, if you set Axis Text Size to 0.08 and the horizontal scale is 50.0, then the text height will be (0.08 X 50) or 4.0.
Horiz Grid Spacing: Specify the distance the vertical lines of the grid will be spaced.
Horiz Text Spacing: Specify the interval that text will be plotted below the grid lines.
Vert Grid Spacing: Specify the distance the horizontal lines of the grid will be spaced.
Vert Text Spacing: Specify the interval that text will be plotted to the left and right of the grid lines.

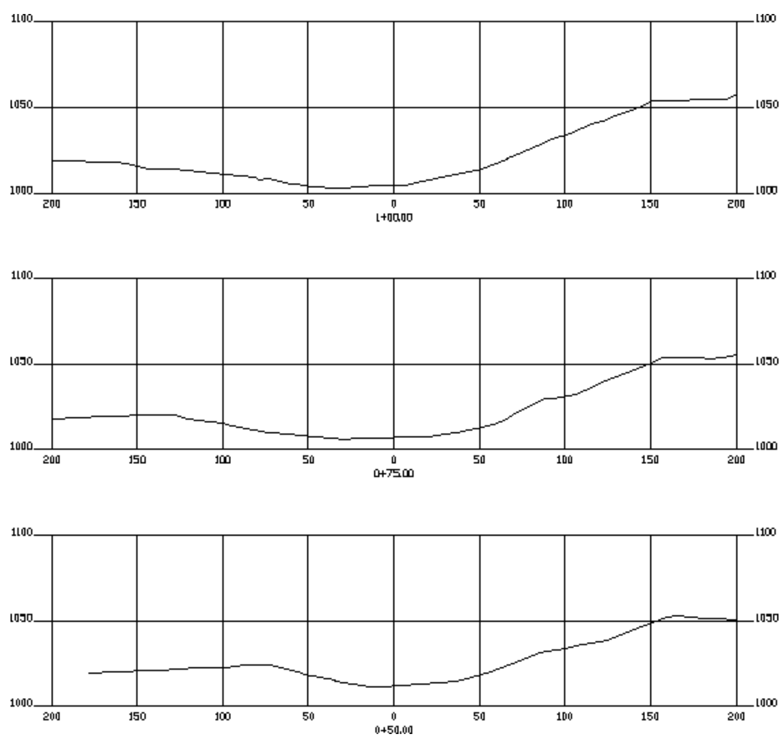
Select the OK button at the bottom of the dialog to begin plotting. For the Vertical Stack and Pick Location options, you are prompted to specify a starting point for the sections. If Sheet option was selected, another dialog appears to specify all the settings for sheet plotting, see details below.

Prompts

Select Starting Point for Row of Sections *pick a point*





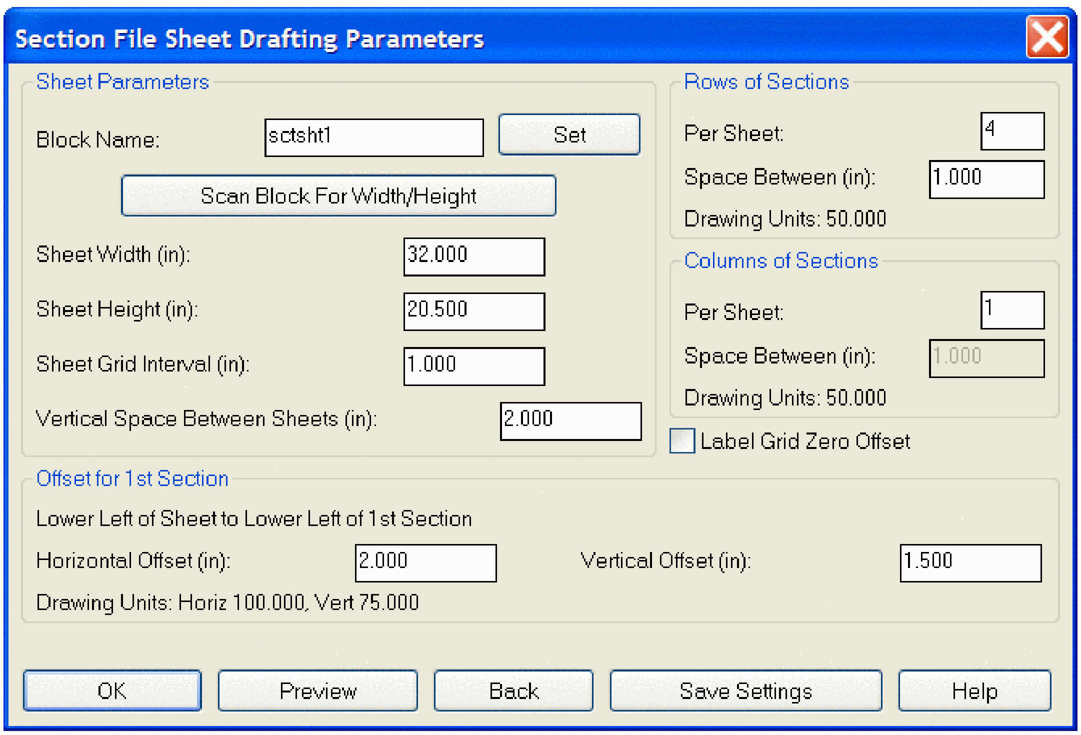


Vertical Stack layout

The Pick Location type of plotting has the following prompts:

Station> 4000.000 Min Elev> 462.849 Max Elev> 472.091
Change datum elev/<Select point that represents 0 offset elev 460.0>: *C*
Starting-Datum Elevation: *450*
The program scans the station data and determines the minimum and maximum elevations, and proposes a datum elevation. If you have pre-plotted a grid sheet and want to reference another local grid coordinate, then change the datum elevation appropriately.
Change datum elev/<Select point that represents 0 offset elev 450.0>: *pick a point*
Station> 4025.000 Min Elev> 463.332 Max Elev> 472.385
Change datum elev/<Select point that represents 0 offset elev 460.0>: *pick a point*

The program continues to prompt until the last station in the range specified is drawn. You can use the Cancel function ([Ctrl] + [C]) to stop plotting, if necessary. If you chose the Vertical Stack option you will be prompted for the starting point for the row of sections. If you selected the Sheets option, and press the OK button, the Section File Sheet Drafting Parameters dialog appears, allowing you to set up how you want the section sheets plotted.



Sheet Parameters
Block Name: Specify the AutoCAD drawing name that will be inserted for each sheet. The default is SCTSHT1 which is included with Carlson 2008, and is stored in the \SUP directory. You can use this or use a sheet block of your own design. The block should be drawn at a 1 = 1 scale since the program inserts it at the horizontal scale setting from the previous dialog.
Distance Between: Controls the distance from the bottom of one sheet and the bottom of the next.

Rows of Sections
Per Sheet: Controls how many sections will be stacked on top of each other on a sheet.
Distance Between: Controls how much space will be placed between the top of the last section plotted and the

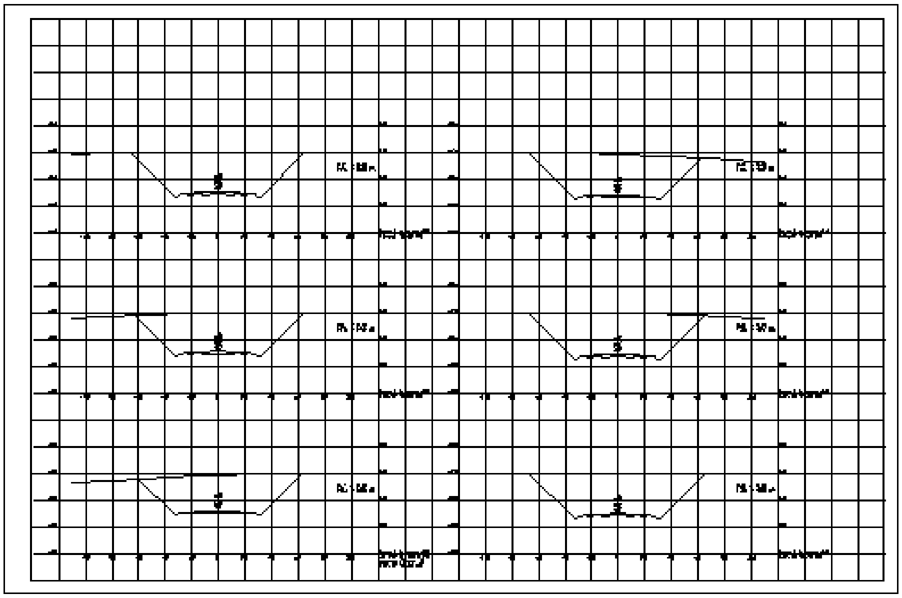
bottom of the next section. The distance between and other values in this dialog are in AutoCAD units. In our above example we are set to 20 horizontal scale so 20 would equal 1 inch when plotted. It is recommended that you set the horizontal and vertical scales in the previous dialog before accessing the sheet parameters dialog so that reasonable defaults will be set automatically.

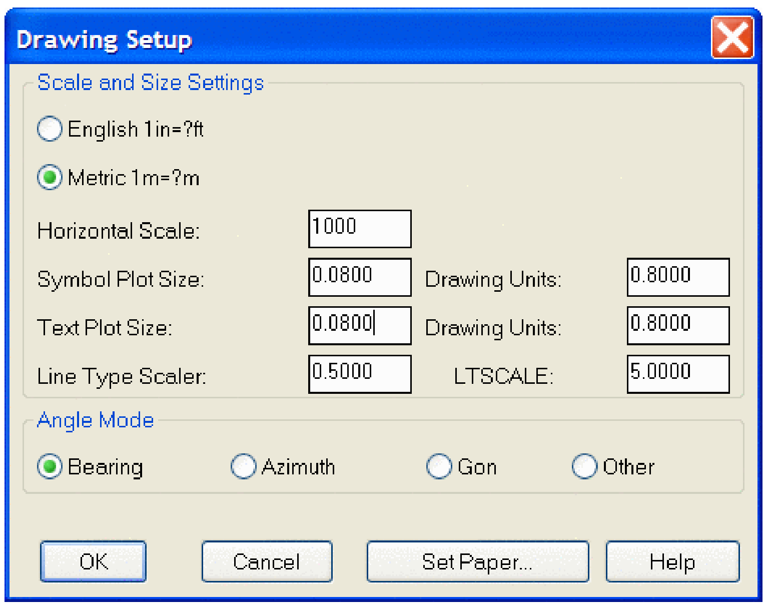
Columns of Sections
Per Sheet: Controls how many rows of sections will be plotted on each sheet.
Distance Between: Controls the distance that the rows of section will have between the centerline of the one section row and the next centerline of rows. This edit box can only be accessed if you have a number of columns greater than one. For example, if you want 15 inches between the columns, specify 300 (15 x 20).

1st Section Offset from
Lower Left of Sheet to CL: X and Y edit boxes allows you to specify where the first section of the first row will be placed relative to the lower left of the section sheet. In our example we specified 160 (8 inches at 20 scale) and 15 (1.5 inches at 20 scale). The Block SCTSHT1 has a half inch border before the 1st grid line and we want to plot starting at the second grid line, which is another inch from the bottom of the sheet. We want the centerline of the first section to be slightly left of the center of the sheet which is 33.5 inches wide so we specify 15 inches (300 at 20 scale).
Previous: This button allows you to return focus to the main dialog and make changes to settings or cancel the program. One thing to remember when plotting sheets with grid lines on them is to switch on the Text Only toggle on so you don't get duplicate grid lines.
Label Grid Zero Offset:
Save Settings: This button allows you to save all the parameters settings to a file so you can easily recall them for another project.
Load Settings: This button allows you to recall the settings saved with the option explained above.

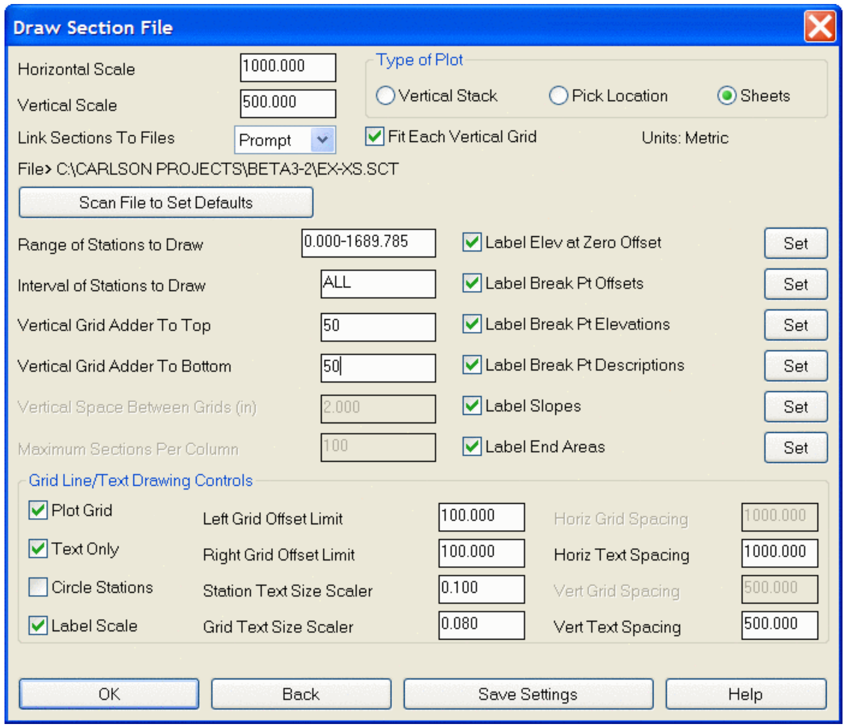
When you select the Layers button this dialog appears allowing you to specify the layer that the files are plotted on. If you are specifying a new layer to create, type the name into the edit box. If you want to select a layer that already exists from the layer list, then click on the Select... button to the right of the edit box. When you select the OK button the program prompts for the starting point for the row of sheets. The default is coordinate 0, 0 though you can select any point you like. With the settings shown in the example dialogs the sections would be plotted as shown below.

Drawing Metric Section Sheets
First, be sure that you are set to metric mode in *Drawing Setup* under the Settings menu. Then set the scales and spacing as shown in the dialog below. This example is 1:1000 scale. When the first dialog is set, click OK to reach the second dialog. There is a different block name for metric sections called schsht2.dwg. This file is located in the Carlson 2008 SUP directory. Choose the parameters for the second dialog as shown. In this case the sheets will have two rows and two columns of sections.

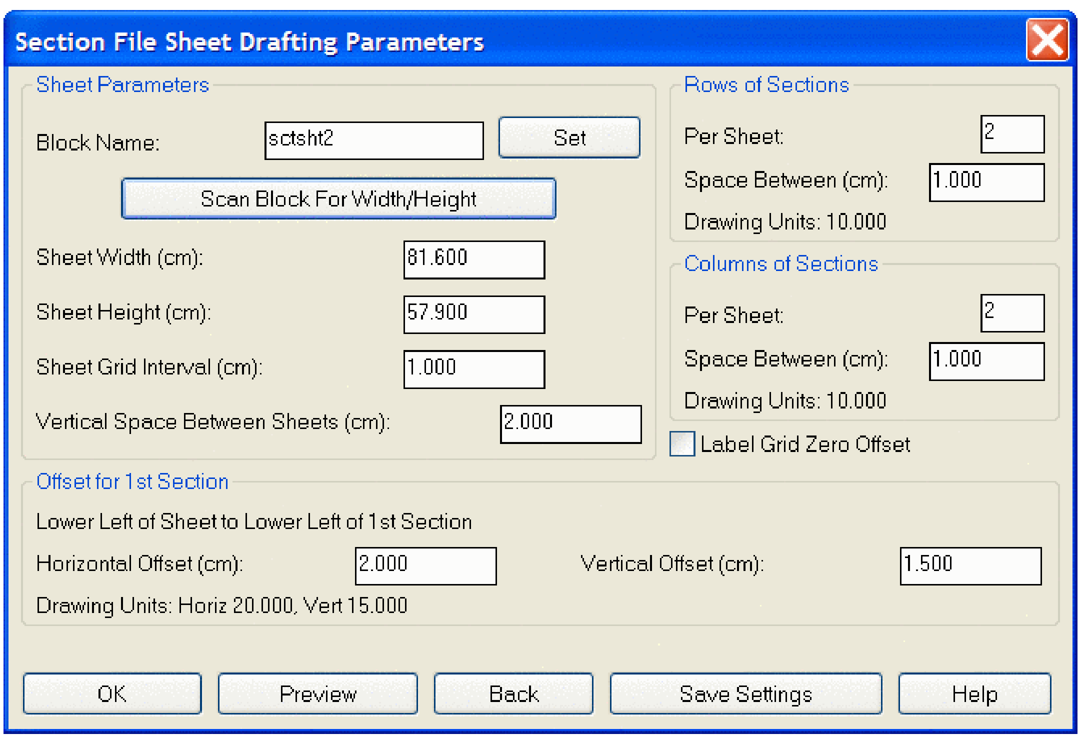




Drawing Setup dialog with metric 1m=?m setting from Settings menu



First dialog with metric settings

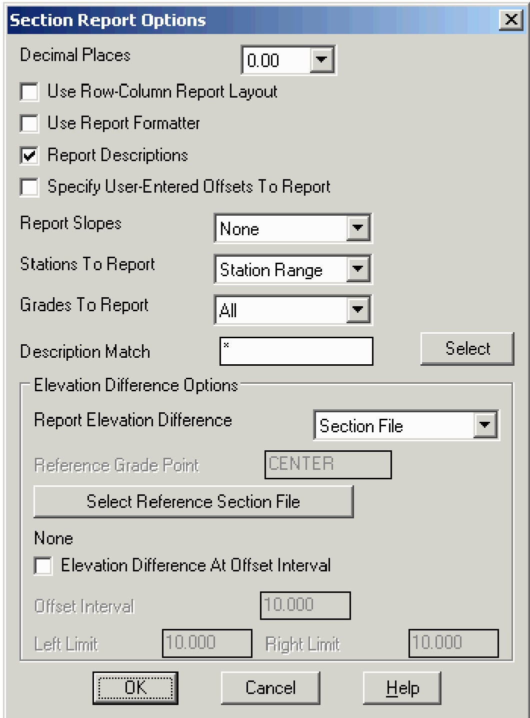


Second dialog with metric settings

Keyboard Command: drawsct
Prerequisite: An .SCT file
File Names: \lsp\drawsct.lsp, \lsp\scadprof.dcl, \lsp\drawsct.arx

Section Report

This command generates a report of a section file for the specified stations. The information contained in the report is determined by the settings in the Section Report Options dialog box.



Decimal Places: Specify the display precision for stations and elevations.

Use Row-Column Report Layout: When checked, offsets are reported in columns. Example reports showing the difference are shown below.

Use Report Formatter: Report output is directed to the Report Formatter which allows for custom reports, as well as being able to export the report to Microsoft Excel or Access.

Report Descriptions: Controls whether the descriptions for each section point are reported.

Specify User-Entered Offsets To Report: After choosing OK from this dialog, the program will prompt for additional offsets to report with interpolated elevations. These are for offsets that don't already exist as section points in the section file.

Report Slopes: Will report the slope between section points. Specify how to report the slopes, either none, percent, ratio, or auto format. Auto format means that slopes less than 10% are reported in percent, while greater slopes are reported as ratios.

Stations to Report: Specify either a range and interval of stations to report or enter each station one at a time.

Grades to Report: This applies to section files that contain subgrades. For these section files, you can

choose which grades to report (top surface or subgrades). All is also an option.

Description Match: This field can be used to filter the section points by their description.

Report Elevation Difference: Reports section elevations by Reference Grade Point, Section File or choose none.

Reference Grade Point: Specify the reference grade ID. Only available if Grade Point option is selected, as mentioned above.

Select Reference Section File: Specify a reference file. Only available if Section File is chosen, as mentioned above.

Elevation Difference at Offset Interval: Used if there is an elevation difference.
The next three options only available if Elevation Difference at Offset Interval is clicked**.**

Offset Interval: Value required.

Left Limit/Right Limit: Values required.

Prompts

Section Report Options dialog *choose options*
Section File to Report dialog *choose existing file*
Starting station for report <0.000>: *press Enter*
Ending station for report <1147.478>: *press Enter*
Station interval (A for All) <100.0>: *press Enter*

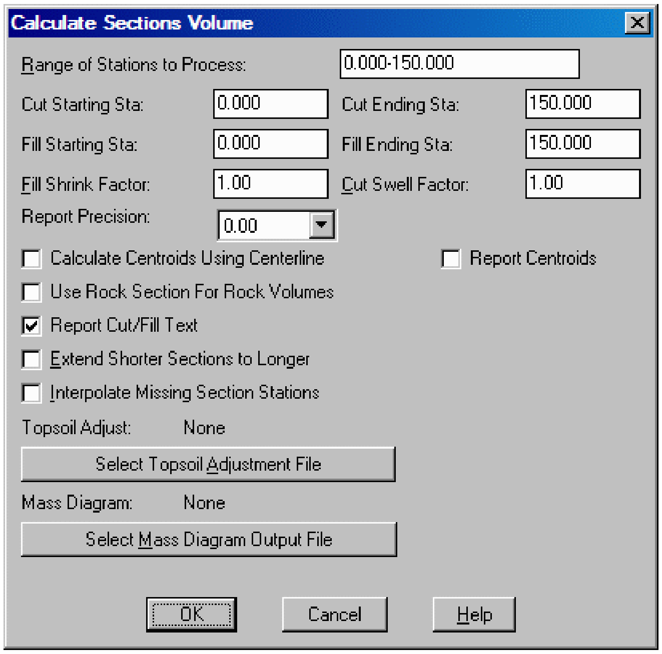
```
=============================================
Row-Column Layout ON
=============================================
Section Report      05/15/2002 18:31
File: C:/scadxml/DATA/125.sct

STATION

16+78.12   -260.00  -259.65  -244.64  -234.32  -213.99
           1401.30  1401.30  1400.92  1400.77  1400.62
           EXTRAPOL 0        0        0        0

=============================================
Row-Column Layout OFF
=============================================
Section Report       05/15/2002 18:31
File: C:/scadxml/DATA/125.sct

STATION: 16+78.12
-260.00    1401.30    EXTRAPOLATED
-259.65    1401.30    0
-244.64    1400.92    0
-234.32    1400.77    0
-213.99    1400.62    0
-209.00    1400.71    0
-185.27    1401.44    0
-183.67    1401.49    0
-183.33    1401.50    0
-158.34    1401.37    0
```

Sample Report

Keyboard Command: sctrprt
Prerequisite: A section file (.sct)
File Names: \lsp\sctrprt.lsp, \lsp\profedit.arx, \lsp\scadewrk.dcl

Calculate Section Volumes

This command will read two section files and compute the cut and fill end areas and volumes. It computes the sections volume in the order they appear in the file. If you need to sort the stations in sequential order use the *Input-Edit Section File* command. Begin by selecting the base section file then the final section file. After specifying the input files the Calculate Section Volumes dialog appears. The settings can then chosen and customized to match your reporting needs. There is an option to apply topsoil removal/replacement adjustments, as well as support for processing sections with subgrades.



Range of Stations to Process: Specify the range of stations to process. Separate stations with a hyphen as shown.
Cut/Fill Starting/Ending Sta.: Volumes are calculated using end areas between the range of stations. Instead of cutting off the volumes exactly at this range, the Ending and Starting Stations for Cut and Fill can be used to have the volume taper from zero at the specified Starting Station to the volume at the first station in the range. Likewise, the Ending Stations can be used to taper the volume from the last station in the range to zero at the specified Ending Station.
Fill Shrink/Cut Swell Factor: Allows you to specify a value that the volume calculated will be multiplied by.
Report Precision: Specify the display precision for the report.
Calculate Centroids Using Centerline: When checked, the program will calculate the centroids using a centerline (.CL) file. you will be prompted to select the centerline file.
Report Centroids: Specify whether or not to report centroids.
Use Rock Section for Rock Volumes: When checked, you will be prompted to select a third section (.SCT) file that will be used to calculate rock quantities.
Report Cut/Fill Text: Specify whether or not to report cut/fill text.
Extend Shorter Sections to Longer: If checked, shorter sections are lengthened to the same left and right offset extents as the corresponding longer sections.
Interpolate Missing Section Stations: If checked, the missing stations are accounted for in the calculations.
Select Topsoil Adjustment File: Specify an optional profile (.PRO) file for haul data output.
Select Mass Diagram Output File: Specify an optional profile (.PRO) file for haul data output.

Prompts

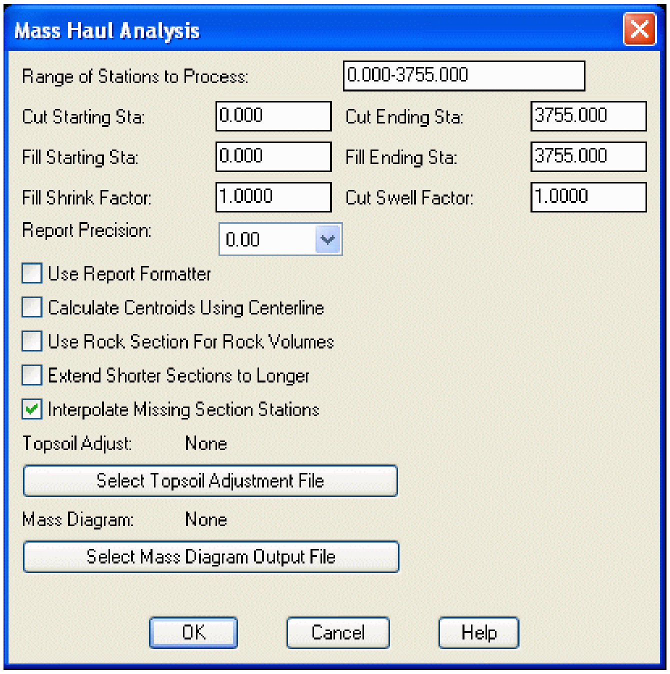
Section File (Existing Ground) to Read *choose existing .SCT file*
Section File (Final Ground) to Read *choose the other existing .SCT file*
Calculate Sections Volume dialog Make selections.

Keyboard Command: calcsct
Prerequisite: Two section (.SCT) files
File Names: \lsp\calcsct.lsp, \lsp\scadewrk.dcl, \lsp\eworks.arx

Mass Haul Analysis

This command will determine the volume and haul distance for each group of net cut and net fill station ranges along a road. The program calculates the optimized cut to fill movements so that the total volume-distance moved is minimized.

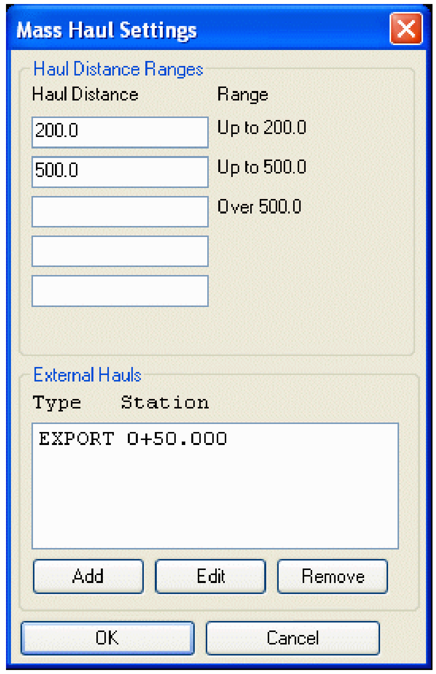
You will first be prompted to select the Existing Ground section file and the Design Surface section file. These two surface files will be used to determine the Mass Haul quantities. If you do not have either of these files, you can create them using the different Create Sections commands under Roads. After you selected your section (.sct) files, the following dialog will appear.



Range of Stations: The program will pick up the range of stations determined by your section files. In this field, you can modify the range of stations to process.
Cut/Fill Starting/Ending Stations: The Cut and Fill Starting and Ending Stations are for tapering the end areas at the start and end of the section range down to zero beyond the station range.
Shrink/Swell Factors: The Shrink Factor is multiplied by the fill quantities and the Swell Factor is multiplied by the cut quantities.
Report Precision: This setting controls the number of decimal places to use in the report.
Use Report Formatter: The Report Formatter will allow you to customize the information reported by the Mass Haul Analysis.
Calculate Centroids Using Centerline: This option will find the center offset for each Cut/Fill area and use a centerline to adjust the station interval along curve segments for the end area volumes.
Use Rock Section For Rock Volumes: This option will use a third section file for reporting rock cut quantities.
Extend Shorter Sections to Longer: This option will find your longest section and match the length of all your other sections to it.
Interpolate Missing Section Stations: Toggle this on to interpolate any missing stations so that the Mass Haul report can use all the stations.
Topsoil Adjustment: This will apply a Topsoil Removal/Replacement definition from the Template Adjustments

to adjust the sections.

Mass Diagram: This will create a Mass Diagram of the cut/fill balance by station. This data is stored in a profile file (.pro) format file, and you can use Draw Profile to draw it.



Mass Haul Settings

The Haul Distance ranges are for reporting the cut to fill volume movements by the different haul distance ranges. The purpose is to evaluate how far the cut has to be moved, and the haul distance ranges can be used to separate the distances for different types of equipment. The External Hauls can be used to specify the stations along the road for borrow pits or dump piles. The program will use volume from these external hauls when the cut/fill of the road does not balance.

Mass Haul Analysis

Existing Section> C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo2-og.sct
Final Section> C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo2-fn.sct
Haul Distance Ranges
Net Cut Net Fill Total In Sta Haul 0 200 Over
Station Station Cut(CY) Import Fill(CY) Export Volume Volume Volume 200 500 500 Avg Haul
0+00.000 1+00.000
1+00.000 2+50.000 1251.729 0.000 1251.729 0.000 1251.729 1127.595 124.134 124.134 0.000 0.000 153.135

12+00.000 10+50.000
13+80.000 12+00.000 887.367 0.000 887.367 0.000 887.367 239.938 647.429 542.651 104.777 0.000 179.887

13+70.000 15+05.340
13+90.000 15+23.200 95.633 0.000 95.633 0.000 95.633 51.559 44.074 44.074 0.000 0.000 137.235

13+80.000 14+20.000
14+20.000 14+50.000 216.434 0.000 216.434 0.000 216.434 96.872 119.563 119.563 0.000 0.000 38.560

14+60.000 14+50.000
14+80.000 14+60.000 43.333 0.000 43.333 0.000 43.333 18.620 24.712 24.712 0.000 0.000 11.818

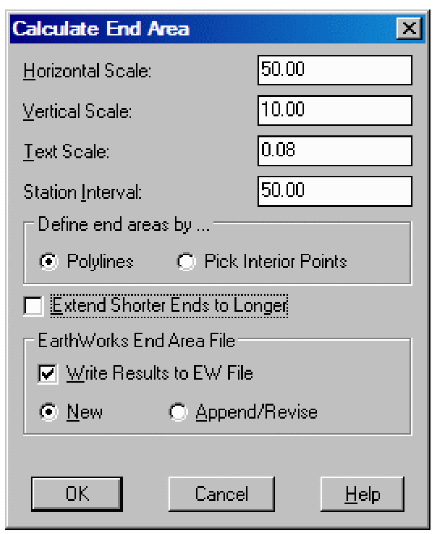
14+70.000 15+00.000
15+00.000 15+14.270 82.194 0.000 82.194 0.000 82.194 29.738 52.456 52.456 0.000 0.000 24.395

Total: 9808.744 0.000 69788.70 0.000 74189.76 66786.40 7403.360 1288.814 930.060 783.423 130.579

Prerequisite: A Section Alignment File and Existing and Road Sections
Keyboard Command: masshaul

Calculate End Area

This command allows the user to select two polylines representing an existing grade section and a final grade section, and calculate the end area. Or you can also specify and define cut/fill end areas by picking interior points. The area calculated can be drawn at a user specified point. Optionally, the command writes the stations cut and fill to an earthwork (.EW) file that can be printed/displayed by the *Print Earthwork File Report* command. This command starts with the Calculate End Area dialog.

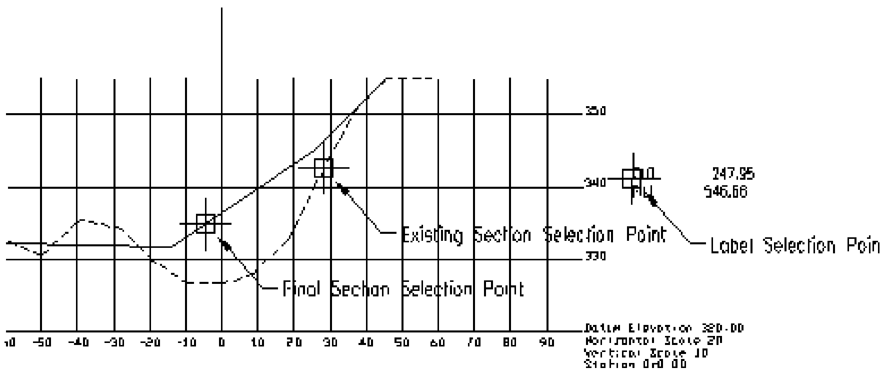


Horizontal Scale: Specify the horizontal scale of the existing cross section.
Vertical Scale: Specify the vertical scale of the existing cross section.
Text Scale: Specify the text size scaler, this value is multiplied by the horizontal scale to determine the final text height.
Station Interval: Only available if *Write Results to EW File* is toggled on. Allows you to specify the station interval that the station prompting will default to as you select the polyline/sections for computation.

Extend Shorter Ends to Longer: Click or leave blank.
Write Results to EW File: When checked, the results will be written to an earthwork (.EW) file. You may create a new file or choose to append/revise an existing file.

Prompts

Calculate End Area dialog *make choices*
Specify Earthworks File (ew) dialog *specify new or existing file* This box appears if *Write Results to EW File* is clicked.
Select existing grade polyline (ENTER to end): *select polyline*
Select final grade polyline: *select polyline*
Calculating End Area...
Cut: 12002.965 Fill: 660.272
Pick Point for Label (Enter for none): *pick point*

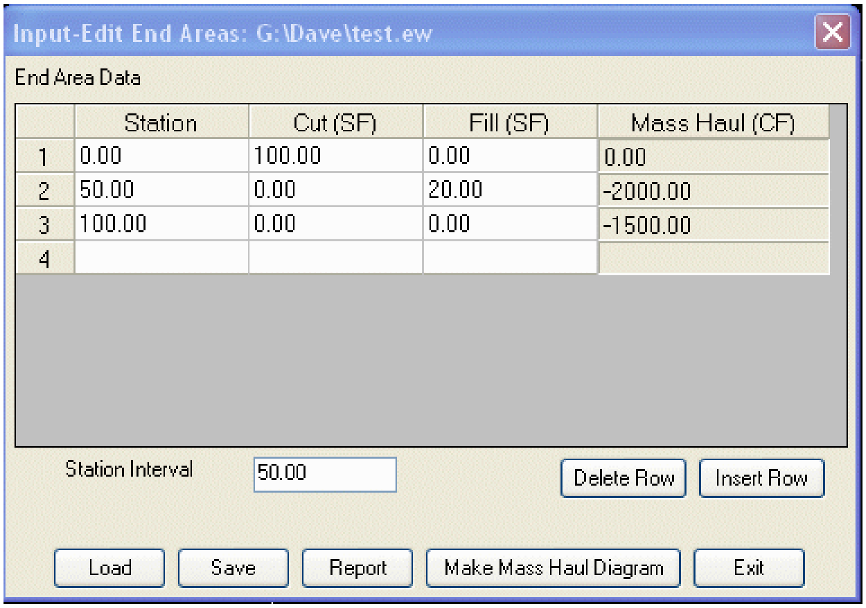


Enter the station <**0.00**>*press Enter* Pressing Enter selects the default station 0+00. If the station does not exist in the file it will be added. If it does it will be revised.
Select existing polyline: *press Enter*
Continue moving along automatically to the next station interval and select polylines. Or enter the station values randomly. The command sorts the .EW file regardless. As a result of this sort feature, the user can select stations in any order and they will be arranged in ascending order for proper volume computation.

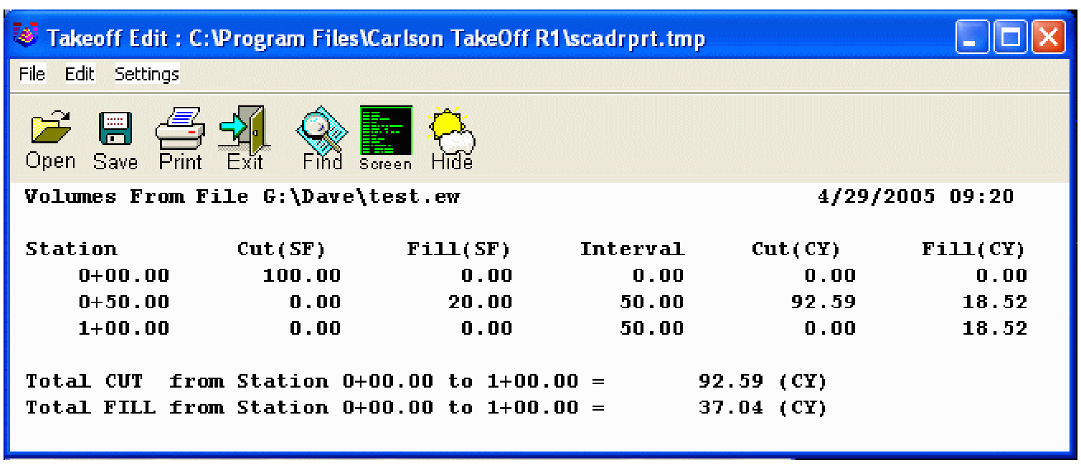
Keyboard Command: endarea
Prerequisite: Plot the existing grade and final grade polyline/section
File Names: \lsp\parea.lsp, \lsp\scadprof.dcl, \lsp\profedit.arx

Input-Edit End Area File

This command allows you to enter in Stations, Cut (SF), Fill (SF) data to calculate a project's Mass Haul (CF).

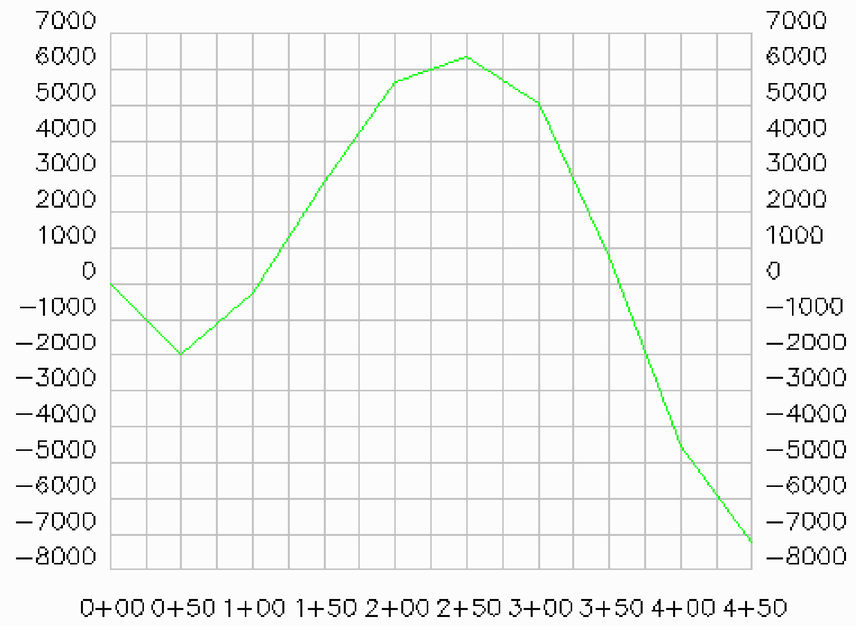


You can set the Station Interval and Delete or Insert a Row at your discretion. The Report button will show you the Cut/Fill at each station as well as the Total Cut/Fill.



Make Mass Haul Diagram

Use the Make Mass Haul Diagram to create a .pro file of your data. Then run the command Report Profile to create a report of your mass haul data or run Draw Profile to create a diagram like the one below.



Prerequisite: End Area Data

Keyboard Command: ewedit

Print Earthwork File Report

This command is used to display/print an earthworks (.EW) file. This file can be generated by several commands such as *Calculate End Area* or *Digitize End Areas.* A standard file selection dialog prompts you for the .EW file, then the volumes report is displayed in the Standard Report Viewer.

Prompts

Earthworks File (ew) dialog *select existing file*
Standard Report Viewer screen appears with volumes.

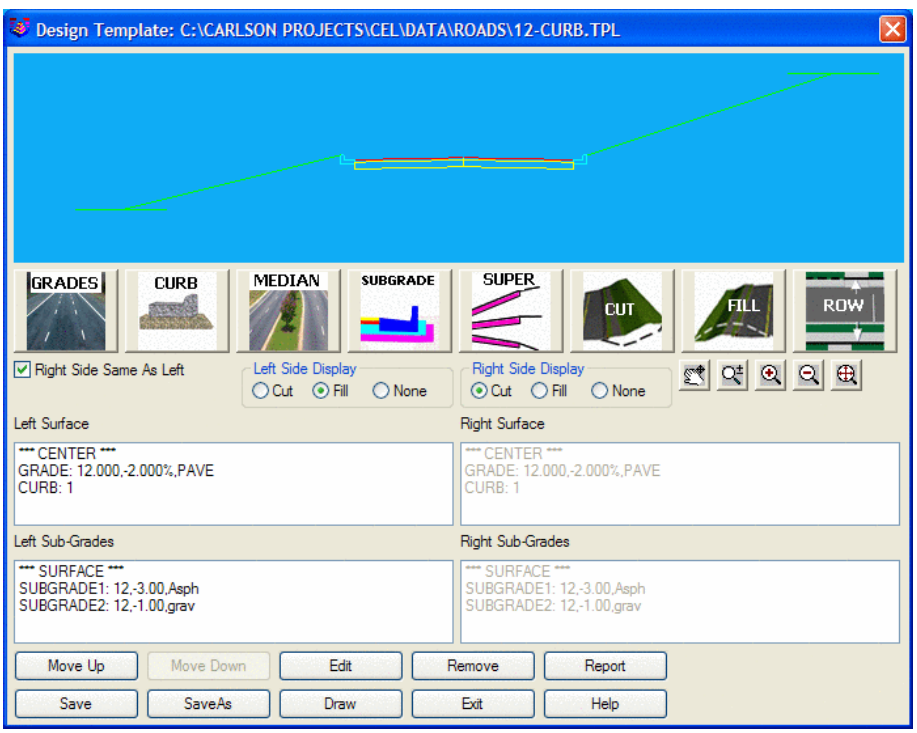
```
 Volumes From File main.ew


 Station      Fill(sf)   Cut(sf)     Interval     Fill(cy)     Cut(cy)
 34+00.00    0.0000     24.7082     0.0000       0.0000       0.0000
 35+00.00    0.0000     75.0246   100.0000       0.0000     184.6904
 36+00.00    0.0000     29.1810   100.0000       0.0000     192.9733
 37+00.00    0.3867      1.4916   100.0000       0.7161      56.8011
 38+00.00   31.1798      0.0000   100.0000      58.4565       2.7622
 39+00.00   57.6465      0.0000   100.0000     164.4931       0.0000


 Total FILL from Station 34+00.00 to 39+00.00 = 223.6657 (cy)
Total CUT  from Station 34+00.00 to 39+00.00 = 437.2270 (cy)
```

S
Keyboard Command: ewreport

Prerequisite: .EW file
File Names: \lsp\eworks.arx

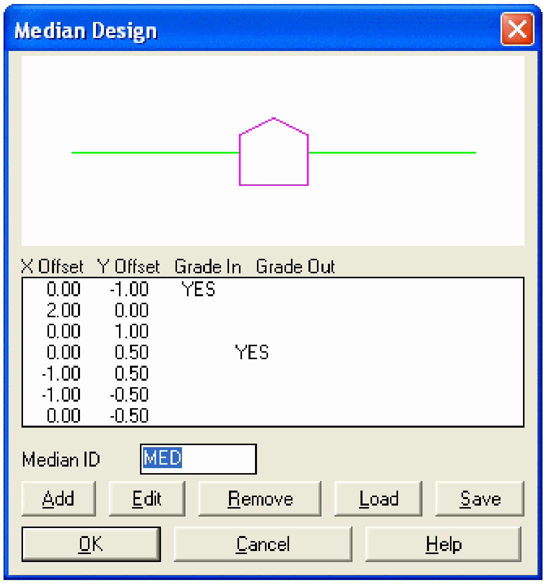
Design Template

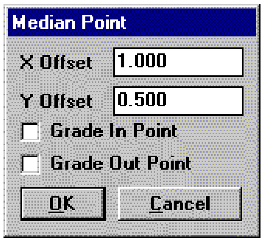
This command creates a template definition file (.TPL file). The template file can then be applied in the *Process Road Design*, *Draw Typical Template*, *Locate Template Points* or *Design Pad Template* commands. The template is designed using the dialog shown below. The top portion shows the template as you create it. In the middle is a row of icons which are the building blocks of the template. They can be chosen in any order by picking on the icon. In the bottom of the dialog are four list boxes that list the elements of the template. The surface elements are listed in order starting from the center. The subgrades are listed from top to bottom order. To add a template element, highlight the position in the list above where to insert the element. Then pick one of the element icons. To change the order of an element, highlight the element and pick the Move Up or Move Down buttons. The Edit button edits the dimensions of the highlighted element. The Remove button erases the highlighted element from the list. There is no limit to the number of surface or subgrade elements. Note that there is a Right Side Same as Left option. When active this option only requires template design for the left side and will automatically mirror the design for the right side.



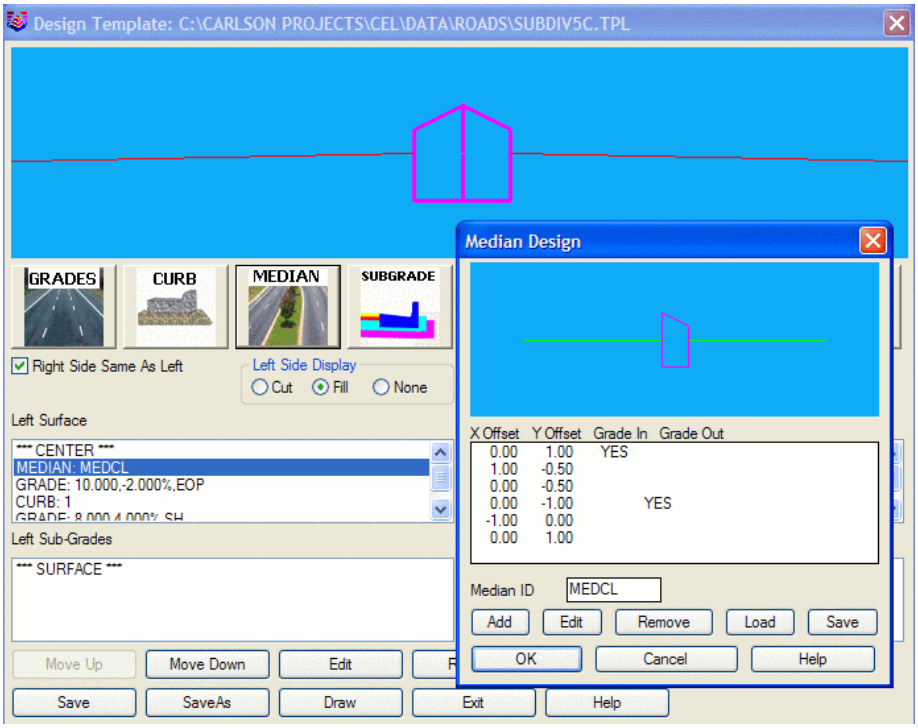
The template surface can be composed of three types of elements: medians, grades and curbs. The median is a flexible closed figure defined in a clockwise direction. Each median point consists of an X and Y offset. The median must be closed and the program will automatically create the closing segment. In the Median Design dialog, the median is shown in the top display and bottom has a list of median points. The display shows the median in magenta and the grade lines in and out in green. For the display the grade in comes from the left and the grade out goes to the right. The median must define the Grade In point which is the point that ties into the incoming surface grade. Also the Grade Out point must be specified for where the surface grade continues out from the medianThese

Grade In and Grade Out points emanate from the starting or "from" position in the coordinate dialog where they are specifiedSince a single median must be placed on the left or right side (and is typically not used symmetrically with right side same as left), you will need to offset the template centerline one-half the median width within the command Process Road Design in order to center the medianYou will also have to move the "C/L" designation, to obtain centering, when using Draw Typical Template.

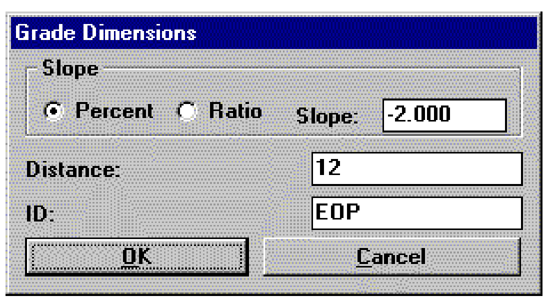




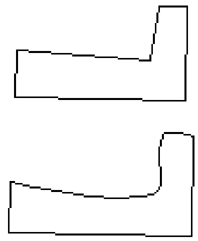
You can design a median for "mirroring" to create a centered effect, as shown below. The only negative to this method is the appearance of a vertical line in the median plot. Medians can be saved and loaded for re-use in other templates.



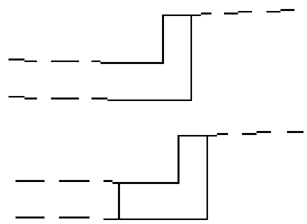
Surface grades can be entered by selecting the Grades icon which brings up the dialog shown. Downhill slopes are negative and the Distance is the horizontal distance. The text ID serves 4 purposes: (1) The ID will be applied as a description to all final template points generated in the form of a coordinate (.CRD) file, (2) The ID can be used as a design point, as in EP+5 indicating 5 feet or meters right of edge of pavement, (3) Points of common ID may be connected by 3D polylines as an output option of *Process Road Design* and (4) Quantities can be generated with reference to the ID and material (gravel, concrete, etc.) entered elsewhere within this command.



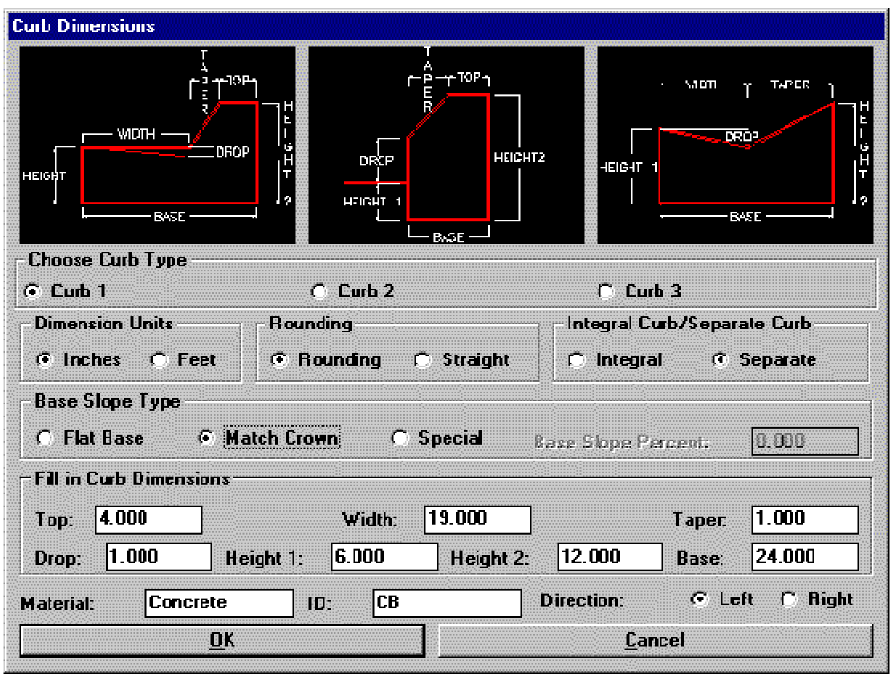
To add a curb, select the Curb icon. The dialog box below appears where you can fill in the curb dimensions. There are three curb types to choose from. The curb dimensions can be specified in feet, inches or meters in metric mode. The Rounding option will smooth the surface of the curb which only shows when the template is applied in commands such as *Process Road Design*. The Integral/Separate option determines whether to draw the front line of the curb to separate the curb from the subgrade. For example, fully concrete pavements that contain a curb would be drawn with the "integral" curb optionThe slope of the curb can either be flat, set to the slope of the incoming grade or set to a user-specified slope. The material name is used in the *Process Road Design* report

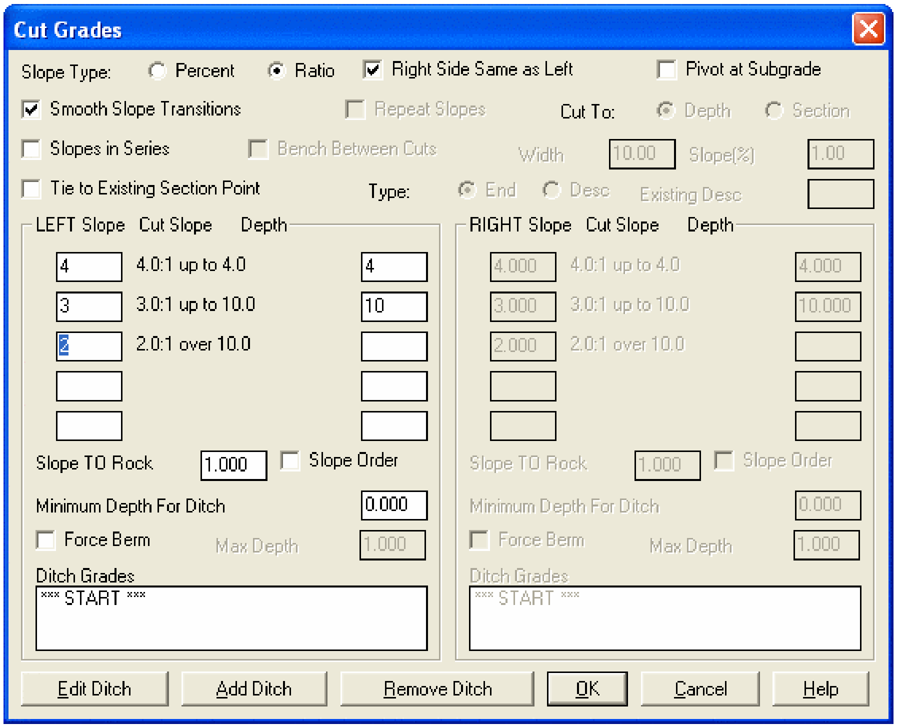


Straight & rounded curbs



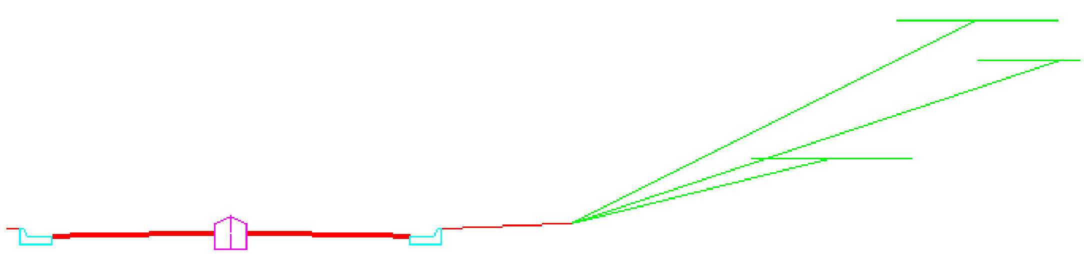
Integral and separate curbs



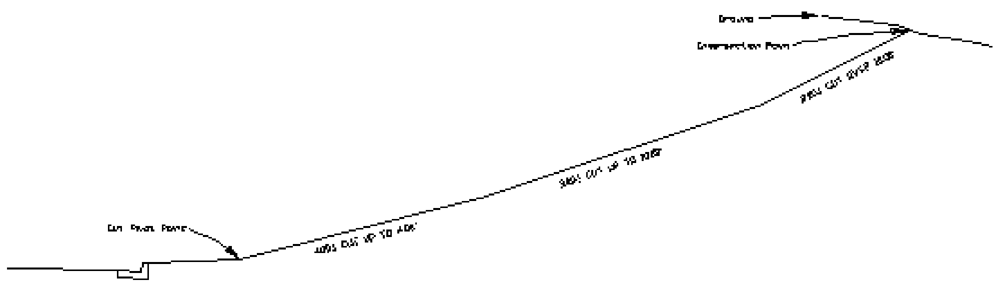


To specify cut treatment, pick the Cut icon. There is room to specify up to five cut slopes which can be slopes in series or slopes to use at different depths. In a simple case of one cut slope, you can just enter the one slope value and leave the depth and other slope boxes blank. For Slopes in Series, each slope is used up to the specified depth until an intersection with the ground. If the intersection is not reached by the first slope, then the next slope continues from where the first ended. If you have more than five slopes, pick the Repeat Slopes option which will repeat the sequence of entered slopes until the ground is reached. The Bench Between Cuts option allows you to enter a bench width and percent slope to be inserted between each cut slope. Besides running the cut slopes to specific depths, the Cut To Section option can be used to have each cut slope intersect a surface from a section (.sct) file. With Cut To Section on, the Process Road Design command will prompt for these cut slope section files. For example, this Cut To Section option could be used when you have a cut bench that occurs at a set elevation but different cut depths as the road profile changes. In this case, you could create a section (.sct) file at this set bench elevation.

With Slopes in Series off, just one of the slopes is used depending on the depth. For example, set the dialog as shown to use 4 to 1 slopes at depths up to 4 feet, 3:1 up to 10 and 2:1 if deeper. The effect is 4:1 if shallow and, by contrast, 2:1 if the fill is deep. The Smooth Transitions option will gradually transition the slopes from one range to the next. In this example, if the depth is 5 feet the slope will be between 4:1 and 3:1. The graphic in the Design Template dialog will explicitly show slopes in series versus individual slope depending on setting (shown below are individual slopes, with slopes in series off):



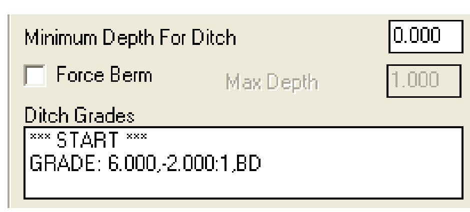
The Pivot at Subgrade option will position the cut pivot point where the bottom subgrade intersects the template grade. The ditch or upslope conditions will then occur from this special subgrade "daylight" pivot point, instead of from the outer shoulder surface pivot point. The Tie to Existing Point will draw the cut slope from the cut pivot point to either the outside offset-elevation or an offset-elevation point with a specified description from the existing section file. This method is used when survey crews take sections and designate the specific slope tie points.

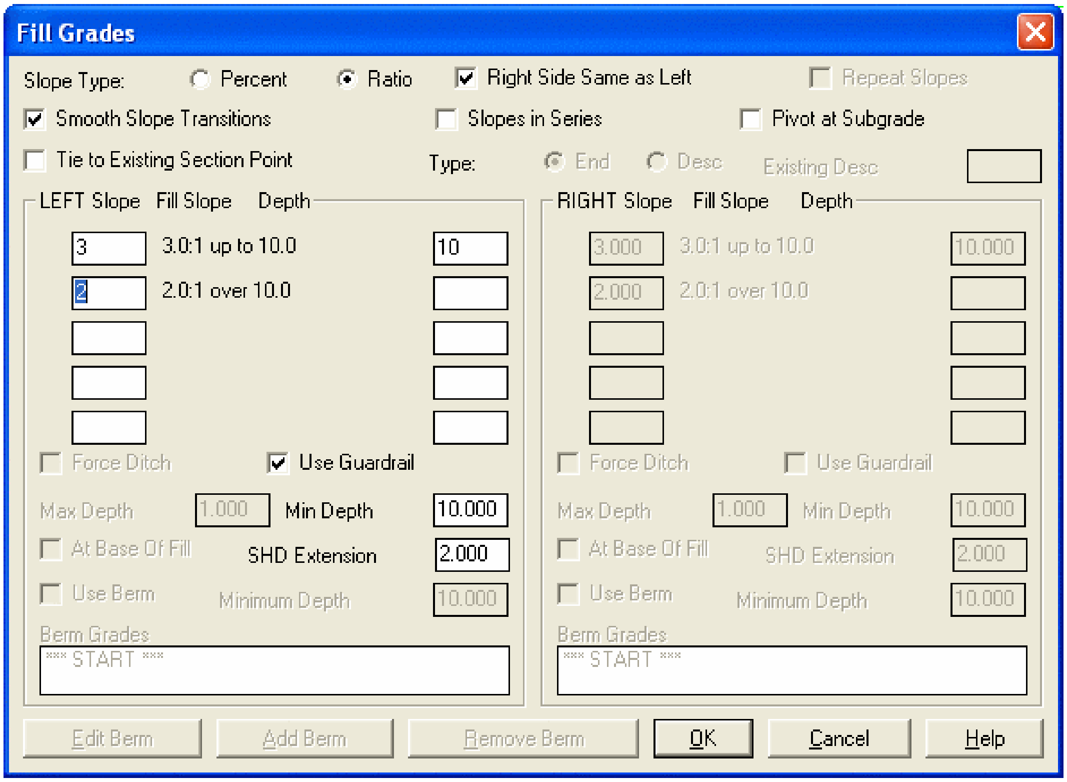


Three cut slopes in series

The Slope to Rock applies in Process Road Design when using a Rock Section File. There are two slope order modes for rock slopes: Slope TO Rock and Slope FROM Rock. For the Slope TO Rock mode, the cut slope will be the Slope To Rock up to the rock surface. After reaching the rock surface, the regular cut slopes apply. For the Slope FROM Rock mode, the regular cut slopes apply up to the rock surface. Then from the Slope From Rock applies from the rock surface to the ground surface.

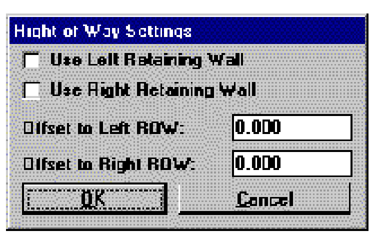
Ditch Grades can be inserted prior to the application of the cut upslope. For curb and gutter roads, there is typically no ditch. But for roads with drainage downhill to the outside and no curbs, ditches are typically used in cut conditions. The Ditch Grades list contains each ditch grade in order from the regular template. Any number of ditch grades can be added by picking the Add Ditch button. To create a V ditch, add just one ditch grade such as slope ratio -1, distance 1. This makes one side of the V. The pivot point for the cut slopes will be the bottom of the V and the other side of the V will be made by the cut upslopes. For a ditch with a flat bottom, you could have two ditch grades such as slope ratio -2, distance 4 and then slope percent 0, distance 2. If a minimum depth for ditch is entered, no ditch will be applied unless the cut exceeds that depth. The Force Berm will apply the Berm (defined using the Fill icon) in cut instead of a ditch up to a certain depth of cut.





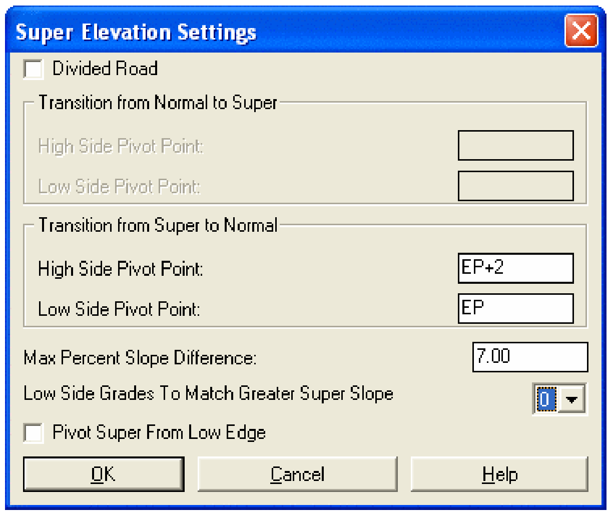
Fill treatment is similar to cut. Up to five slopes for different depths can be specified. Slopes in Series and Smooth Transitions work the same way as cut. Berm Grades are the fill equivalent to Ditch Grades. Fill treatment does have some extra options. Guardrail Expansion will extend the last template surface grade the specified Shoulder Distance when the fill is greater than the Min Depth. The Force Ditch option has two different methods to apply the Ditch Grades from the cut definition. With "At Base Of Fill" on, Force Ditch creates the ditch where the fill slope hits existing ground. With "At Base of Fill" off, the Force Ditch method applies the ditch grades from the template pivot point. The Minimum Depth for Berm Grades will only draw the Berm Grades when the fill depth is greater than the specified value.

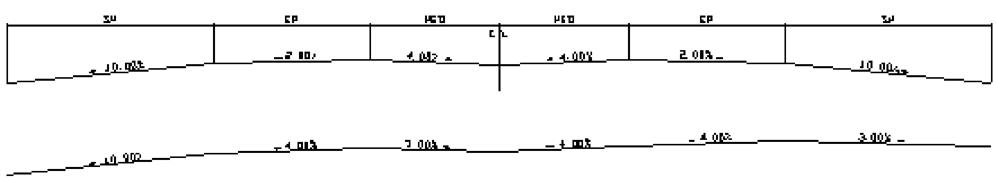
The Right of Way icon brings up the dialog shown which allows you to specify whether to use a retaining wall to keep the cut/fill slopes from crossing the right of way. The right of way data is stored in a centerline file (.cl file) as stations and offsets for the left and right sides of a centerline. When the retaining wall option is active, the cut or fill slope will go at the design slope up to the right of way and then the slope will tie into the ground by going straight up or down. Without the retaining wall option, the cut or fill slope will become steeper in order to tie into the ground at the right of way. For example, if the cut slope is 50% but this slope ties into the ground past the right of way, then the slope will be modified to something steeper such as 65%. The Offset ROW options will force the tie in the offset distance before the right of way.



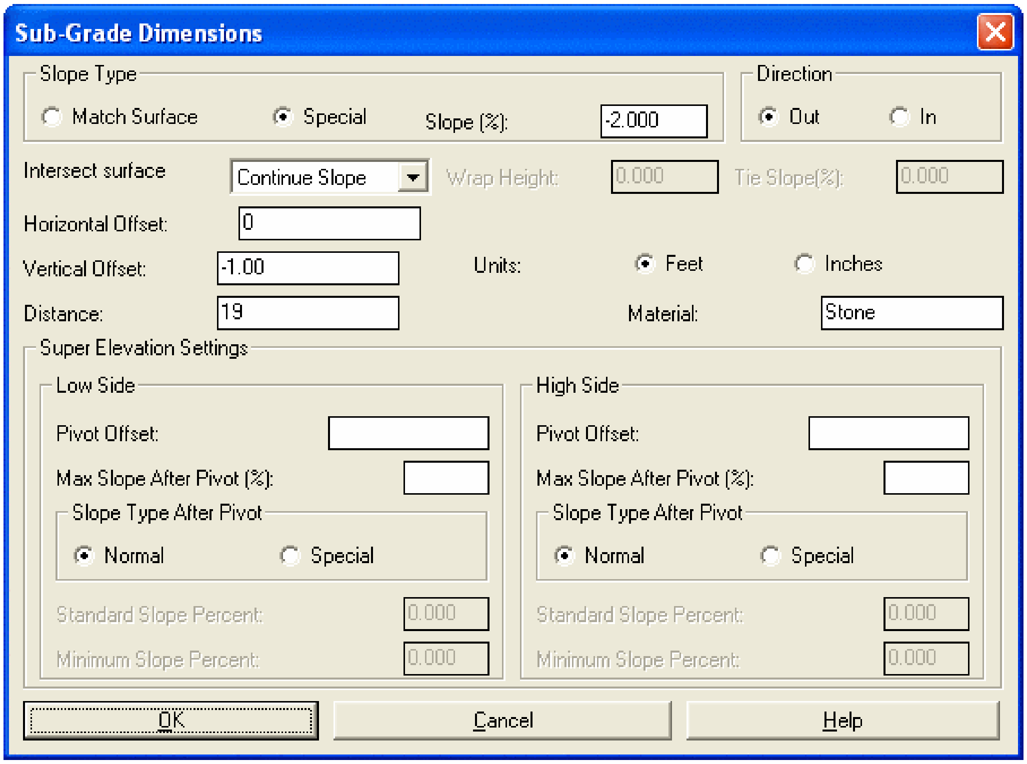
The Shoulder Super Elevation icon specifies where on the template the slopes will transition between super elevation slopes and normal slopes. The transition point is identified under Pivot Point by the template id for the grade, curb

or median. Note that the pivot point can be specified as an ID plus a distance as in "EP+2". Starting from the center, the template grades will be in super up through this template segment. For example, based on the template shown in the first dialog of this command, the EOP Pivot Point the Super Elevation Settings dialog will create the first EOP grade in super while the curb and grade S will be at normal grade. The High and Low Pivot Point options allow for different transition points depending on which side is raised by the super elevation. The Max Percent Slope Difference is the maximum difference between the super elevation grade and the normal grade at the pivot point. For example with a Max Percent Slope Difference of 7%, if the super elevation grade is 6%, then the slope after the pivot on the high side will be -1% even if the normal design slope is steeper than -1%. If the grades do not start from the center in super, then the Divided Roads option can be used. With this option, the grades start from the center as normal and then transition to super at the Normal to Super Pivot Point.



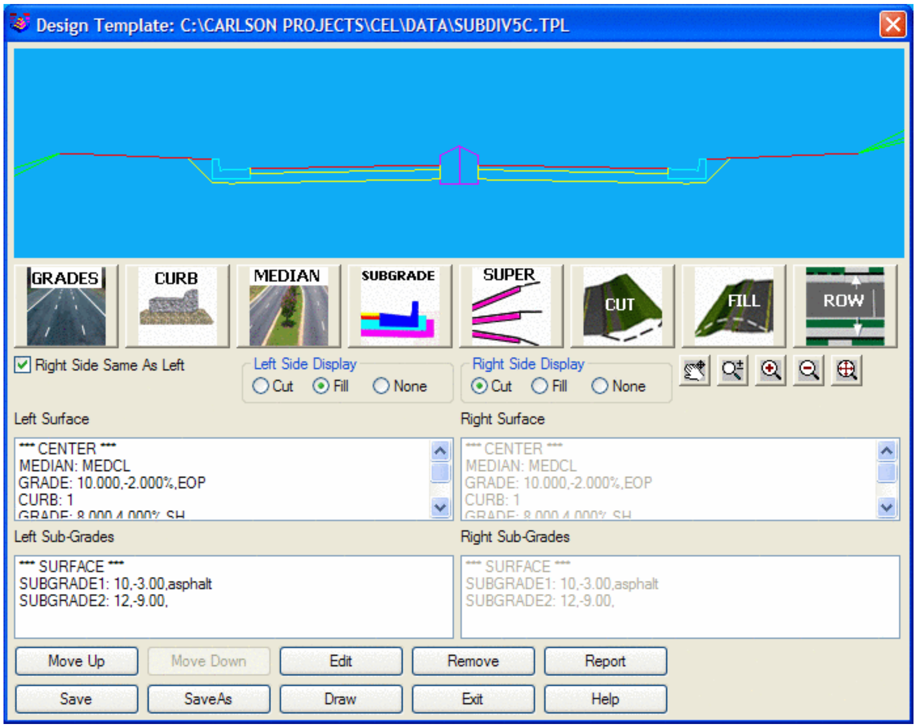


Example of super elevation of 4% to the right for a divided road with a Max Difference of 7%. The normal template is shown above. The Normal to Super Pivot Point is MED and the Super to Normal Pivot Point is EP. The result is that the EP segment is in super and the SH and MED segments are at normal slope. On the left, the SH segment is at the normal -10%, the EP segment is at the super elevation slope of -4% and the MED segment wants to be at 4% but ends up at 3% because this meets the Max Difference requirement. On the right side, the MED segment starts at the normal -4%, then the EP segment transitions into the super -4% and then the SH transitions back to normal which results in a 3% slope because of the Max Difference requirement.

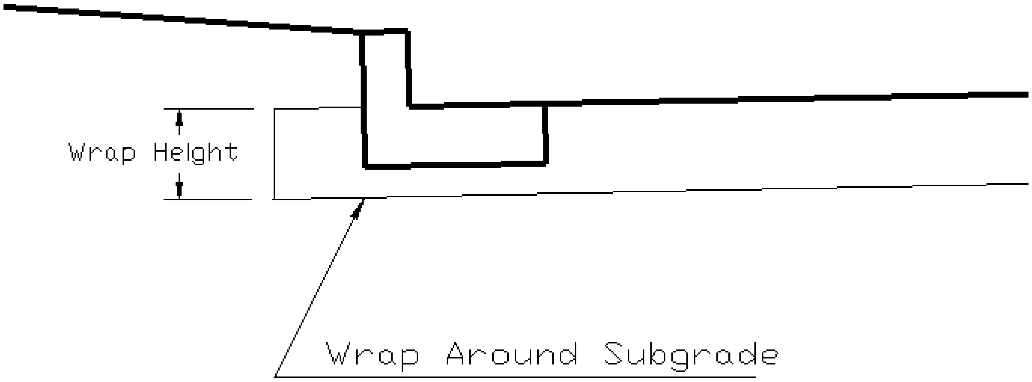


To add subgrades click the SubGrades icon which brings up the dialog shown. The subgrades are areas below the template surface. There can be any number of subgrades stacked one below another or side by side.

The subgrade starts from the surface at the distance from the center set under Horizontal Offset. To start from the centerline, enter 0 in Horizontal Offset. First the subgrade moves straight down from this Horizontal Offset. The depth down is specified in Vertical Offset in feet units or meters in metric mode. The Vertical Offset normally should be set as a negative number. The bottom of the subgrade then either moves away from or towards the center depending in the Direction In or Out setting. The distance to move is specified under Distance. The Slope Type for the subgrade bottom can be either set to a specified slope or set to match the grades of the surface. After moving the specified distance, the subgrade will tie back into the template surface either by going straight up, by continuing at the subgrade slope until intersecting the surface or by wrapping around. The commonly used "continue slope" approach will extend the slope until it hits something (like a curb or another surface segment). It will not trim. So if the pavement segment is 12 feet to a curb, it is better to enter 10 and "continue slope" than to enter 12 exactly, as a "tilted" curb may place the curb edge at 11.98' from the start of the subgrade, causing the subgrade to go past face of curb and intersect back of curb. Also, for angled tie-ins of subgrade from base of curb to the surface, such as the example shown below, be sure the distance entered is less than what would intersect the surface, so that the "extend" effect will create the intersect. In this example, the first subgrade (asphalt) is "continue slope", the second (gravel) is "straight up" and the third (gravel tie in behind curb) is "continue slope".



The Material field is an optional description that is used in the *Process Road Design* report. Special super elevation pivot points may optionally be specified. The values for Horizontal Offset, Distance and Pivot Offset can be specified by template ID. For example, EP could be used in Distance to have the subgrade have a width of the EP grade. Also expressions can be used such as EP+5 to go the distance of the EP segment plus 5. This is especially useful for template transitions so that if the EP grade varies the subgrade width will automatically adjust.



Example of Wrap Around Subgrade

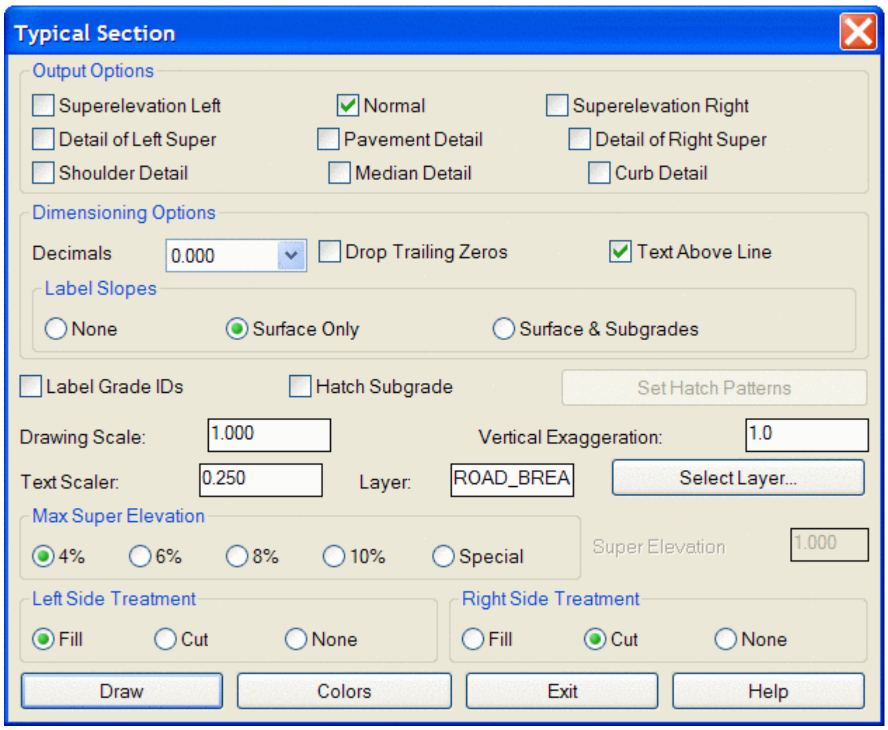
Keyboard Command: template
Prerequisite: None

File Name: \lsp\tplmake.arx

Draw Typical Template

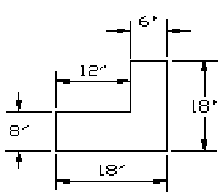
This command draws a template and labels the slopes and distances. The cut and fill treatment can be shown on the left and/or right sides. All the cut/fill slopes are shown for the different depths when multiple slopes are defined. There are options to draw the normal template, super elevation or details of different sections.

You will be prompted to select the template (.TPL) file first, then the Typical Section dialog appears. Specify the parameters and press the Draw button.

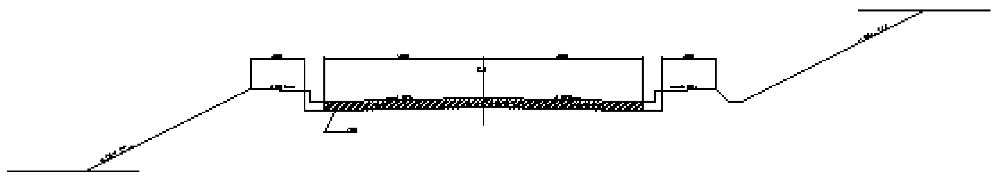


Prompts

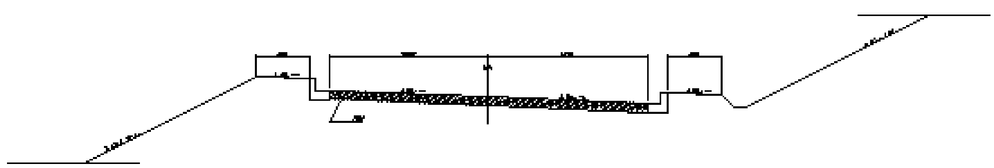
Template File to Read Specify a template file.
Typical Section dialog Set your options then click Draw.
Pick Starting Position: *pick a point*



Curb Detail



Normal Typical Template



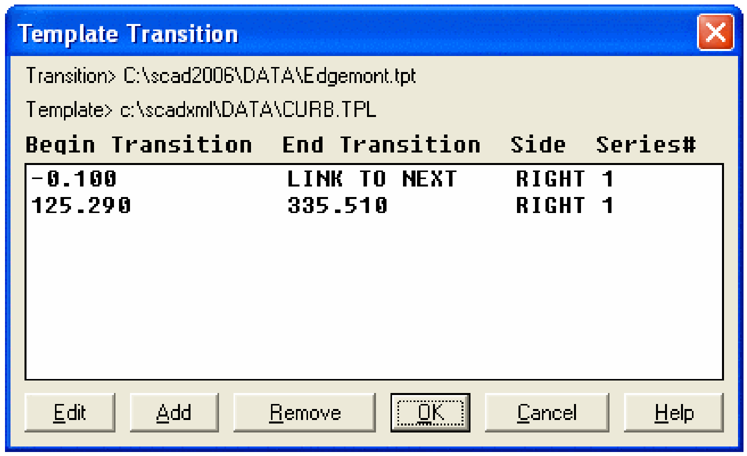
Typical Template with Left Super Elevation

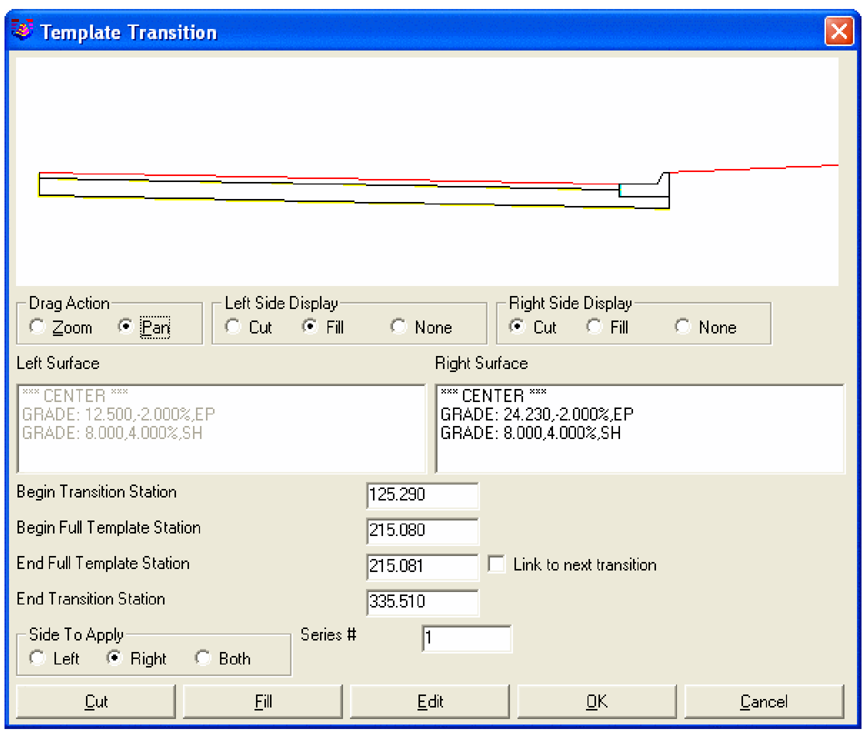
D
Keyboard Command: typical
Prerequisite: A template file (.TPL file)
File Name: \lsp\eworks.arx

Template Transition

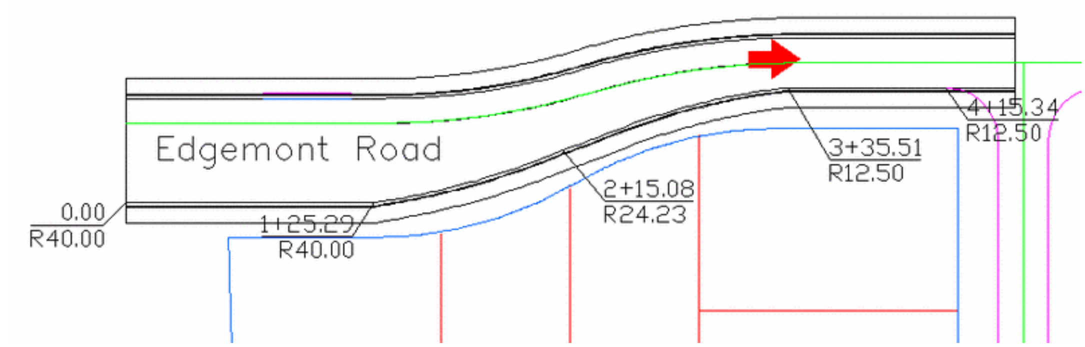
This command creates a template transition file (.TPT file) that can be used for the commands *Locate Template Points* and *Process Road Design*. The template transition is associated with a typical template (.TPL) file. The template transition file defines changes in grade distances or slopes for a specific template ID through a specified range of stations. Lane widths, for example, can be made to expand and contract. You can only modify existing template grades. Template Transition does not allow curbs, medians, subgrades or cut/fill treatment to be modified. Also new template elements cannot be added and existing elements cannot be removed. For this reason, lanes of road that "emerge" and slope distinctly from standard road lanes would need to be entered as small (0.001 in width) segments in the original template, available for expansion using Template Transition. Template Transition offers one of 3 ways to change template widths and slopes. Another way involves use of Template Point Profile and Template Point Centerline, where a particular template ID can be directed to follow a specific profile and centerline of its own. The third method is template-to-template transitions using Input-Edit Template Series, where distinct templates transition one to another. All three methods require that template IDs "pre-exist" in order to be expanded, or to follow profiles and centerlines, or to transition between template files. So the technique of making very short

phantom segments for emerging and disappearing "lanes" or roads with distinct grades is universal. If special slopes are not involved, lanes can expand and contract without creation of phantom segments in the original template. Only clever use of Input-Edit Template Series, where templates with no curbs could "end" and templates with curbs can begin at specified stations, can effectively make "new" features like curbs and medians materialize.





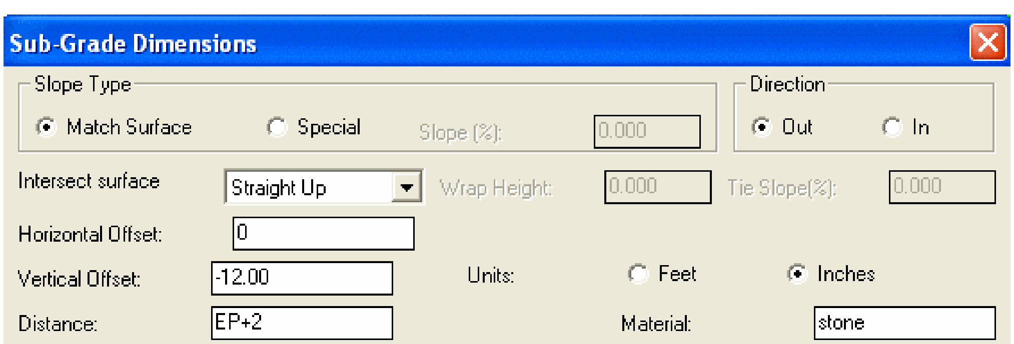
Reviewing the below plan view, when you are given stations and offsets that define a template position like edge-of-pavement (above), you can use Template Transition effectively.



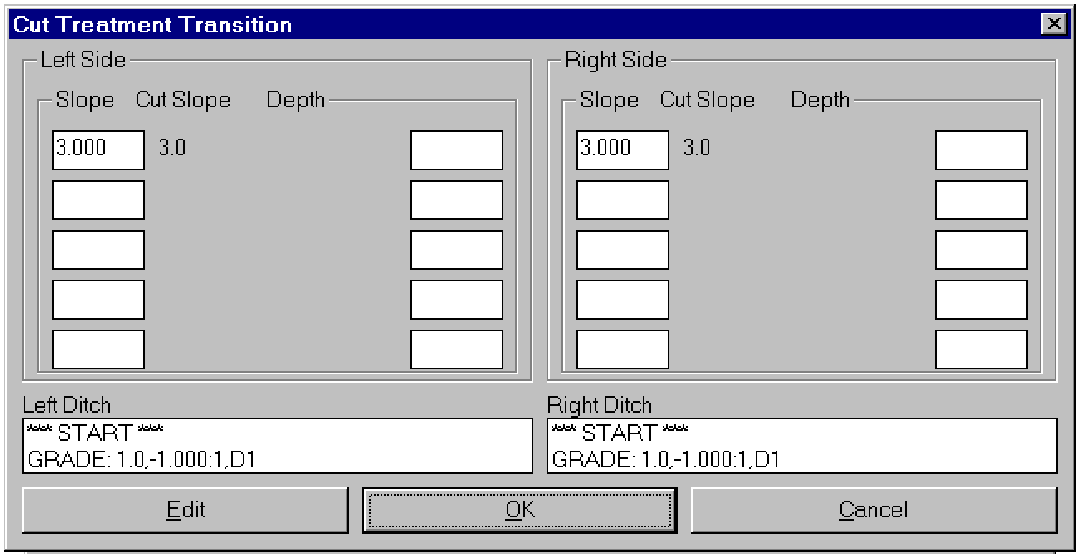
The first Template Transition dialog shows a list of the transitions, covering the above right-lane variable width. To add a transition, click the Add button. This brings up the second Template Transition dialog which shows the transition template for the second segment. The middle sections list the template grades that can be changed. To modify a grade, highlight the grade and click the Edit button.

The Begin Transition Station is where the normal template begins to transition to the modified template. The Begin Full Template Station is where the modified template is used entirely. The End Full Template Station is where the template starts to transition back to normal. The End Transition Station is where the template has returned to normal. This method is designed for elements like passing lanes which expand from normal then contract back to normal. But you can also use this method for roads that start off or end expanded or altered. For example, to start off the road at a 40' edge-of-pavement dimension, it is necessary to transition up from 12.5' (normal dimension). If you need to have 40' at station 0, then enter station -0.01 as the "Begin Transition Station", and enter station 0 as the "Begin Full Template Station". Select the EP grade in the dialog, and change it to 40'. Then click "Link to next transition". The Link to Next Transition option joins the current transition to the next transition without returning to the normal template. This takes you to the second dialog, shown above. You sustain the 40' width from Begin Transition Station 125.29 and transition at station 215.08 to a 24.23' dimension. Then quickly end the transition at station 215.081 for the "End Full Template Station". Finally, transition back to normal 12.5' by entering 335.51 for "End Transition Station".

There is another "trick" to using Template Transition with templates that include subgrades. The subgrades will not automatically extend and follow the expanded grade IDs such as EP for "edge-of-pavement", unless the subgrades are defined in terms of the IDs themselves within Design Template. Subgrades that expand "at slope" to intersect a curb, for example, can expand naturally as the curb position moves outward on the right side. But subgrades that go "straight up" at back of curb at offset 14.5' in this example will stay at 14.5', unless defined as shown below by referencing the "EP ID:



Cut and Fill slopes can also be transitioned by picking the Cut and Fill buttons. Ditch and Berm grades can also be modified here.



Transitions can also be applied to the left, right or both sides. This allows you to have separate overlapping transitions for the left and right sides.

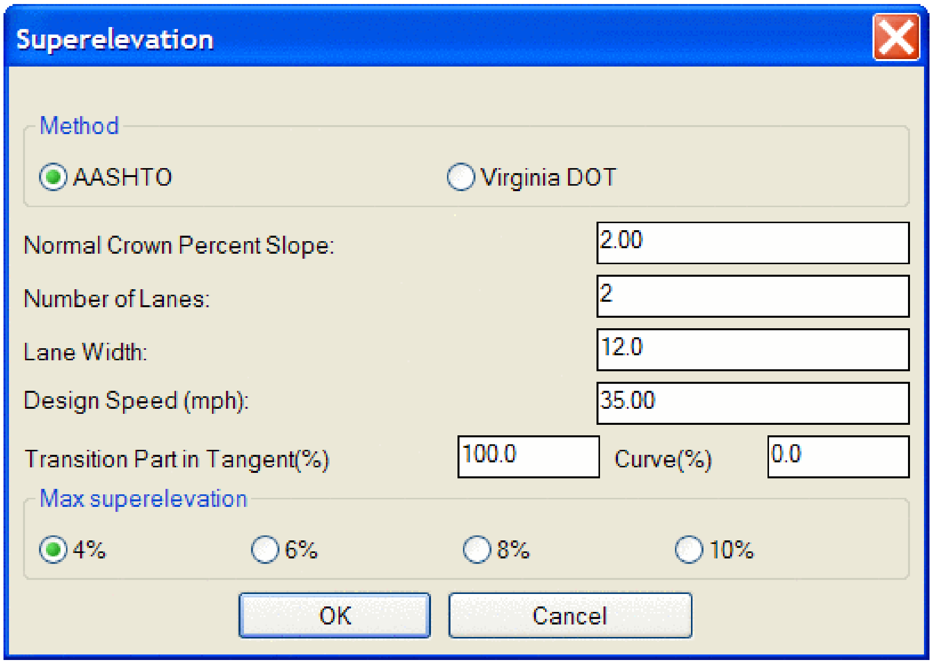
Prompts

Template Transition to Edit/Create Choose New to create a transition file or Edit to modify a transition file
Template File to Edit: Specify a transition file
Template Transition dialog

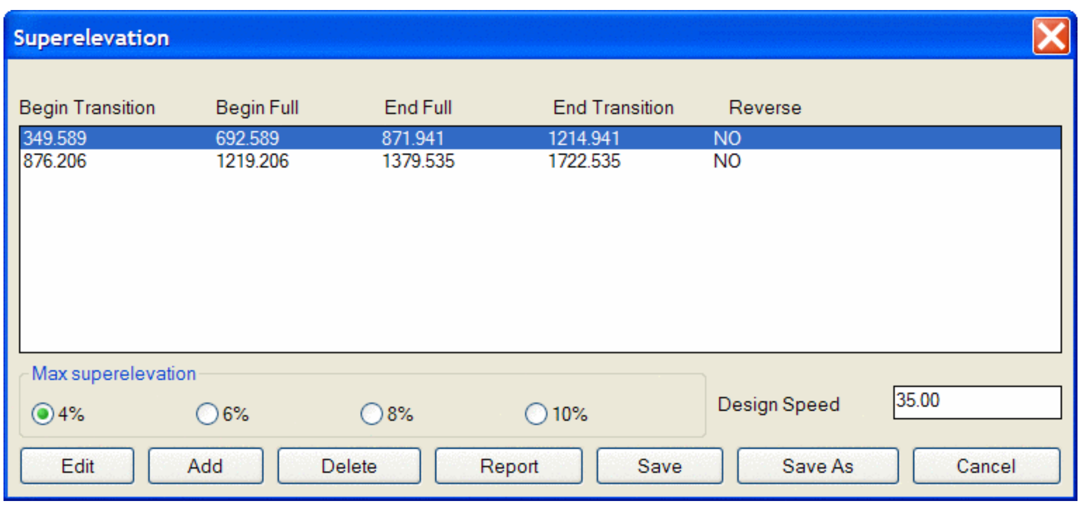
Keyboard Command: tpltrans
Prerequisite: A template .TPT file
File Name: \lsp\tplmake.arx

Input-Edit Super Elevation

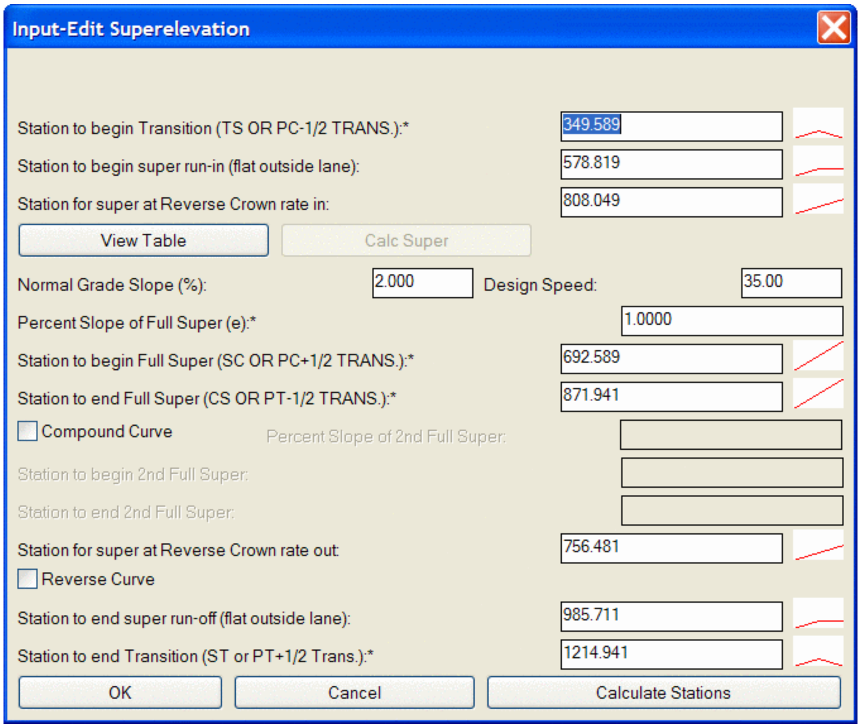
This command is an editor for super elevation stationing. The super elevation data is stored in new or existing super elevation (.SUP) files. When creating a new super elevation file, there is an option to read a centerline file and build the super elevation stationing based on the curves and spirals in the centerline using AASHTO-based stationing or optionally, the Virginia DOT method. The AASHTO calculations are based on the equations in chapter 3 of the 2004 Green Book titled Geometric Design of Highway and Streets. The length of the transition from normal crown to superelevation will be automatically computed by the program using either method based on the design speed and other settings, but the user can control what percentage of this transition to and from superelevation occurs in the tangent leading up to the curve or in the curve itself.



The main superelevation dialog displays a list of each super elevation transition. These entries should be sequentially entered from lowest to highest stations. To edit the super elevation stationing, highlight the entry line and click Edit. The Add button creates a new entry below the current highlighted row or at the top of the list if no row is highlighted. The Delete button removes the highlighted row from the list. The Save button saves the super elevation file. To exit the program without saving, click the Cancel button.



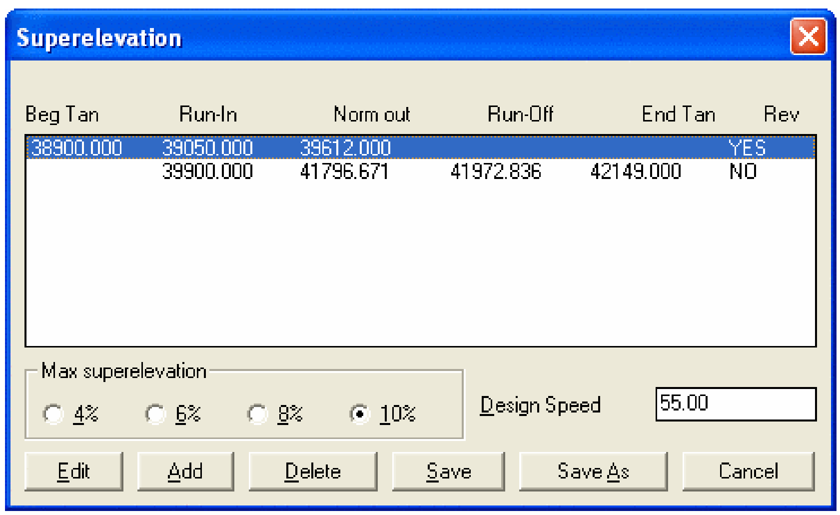
The super elevation stationing is entered in the Input/Edit Superelevation dialog. The View Table button shows a table of the super elevation slope for the delta angle and radius at different design speeds. The Calc Super button calculates the slope of full super given the design speed. The station entries are defined as follows:



Station to begin transition: where normal crown rate begins to transition
Station to begin super run-in: where slope becomes flat
Station for super at normal crown rate in: where slope equals negative of normal crown rate
Station to begin full super: where slope reaches full super slope
Station to end full super: where slopes begins to transition from full super back to normal
Station for super at normal crown rate out: where slope equals negative of normal crown
Station to end super runoff: where slope becomes flat
Station to end transition: where slope returns to normal crown rate

Given these various Station settings, an unequal rate of change can occur between any two stations. However, the program can calculate the stations to set an even rate of transition, as long as it knows the max superelevation, the normal crown slope and the station to start transition, start full super, end full super and end transition. The Calculate Stations button therefore calculates the stations for begin run-in, normal crown rate in, normal crown rate out and end super run-out. To calculate these stations the values with an "∗" must be entered.

The Compound Curve option allows you to specify a second superelevation slope for a compound curve. In addition to specifying the second slope, the starting and ending stations for this slope must also be entered. The Reverse Curve option is similar to the Compound Curve option. A typical Reverse Curve is shown below in plan view and as it would appear in the summary dialog:



Station 399+00 is the "pivot" where superelevation left flattens and turns into superelevation right.

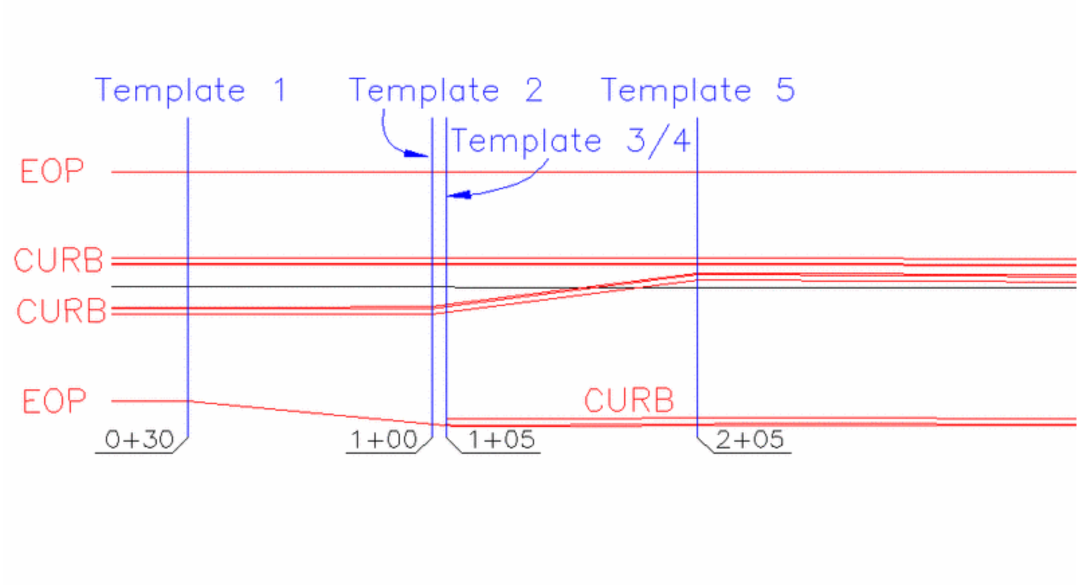
Prompts

New or Existing Super Elevation File dialog
Superelevation File to Process Specify a superelevation file.
Superelevation Editor dialog

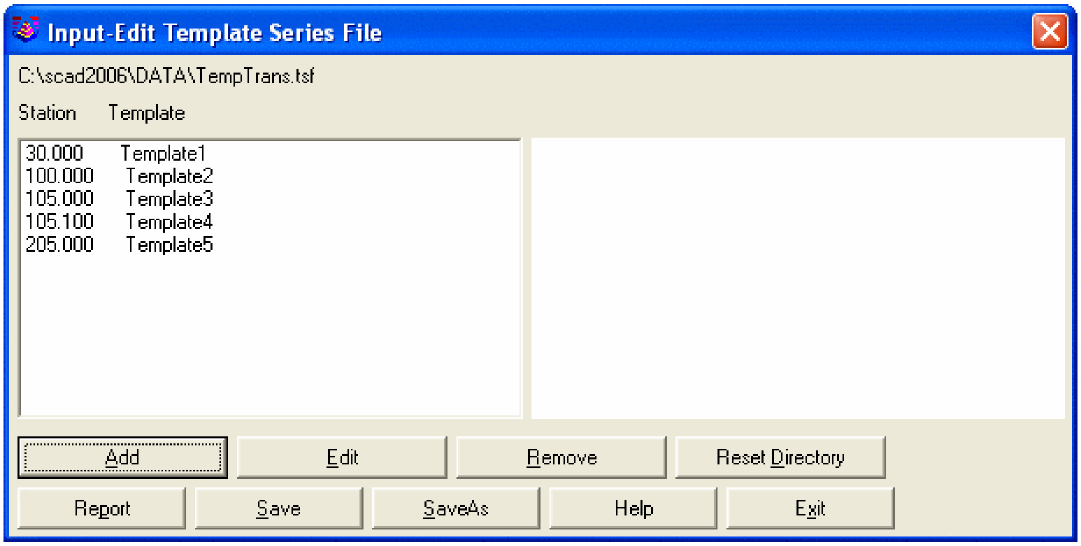
Keyboard Command: super
Prerequisite: None
File Names: \lsp\eworks.arx, \lsp\makeesup.lsp

Input-Edit Template Series

This is another method of widening lanes or causing templates to change: direct template-to-template transitioning. Using this command, you specify the station where one template "ends" and the station where another template "begins", and the program auto-transitions between templates. For the transition to work optimally, the templates should share the same IDs. If the templates are distinct with separate, unrelated IDs, then by ending template1 at station 500 (for example) and starting template2 at station 500.01, a very abrupt transition can be accomplished. For modifying templates, the Template Series method is an alternative to both the Template Transition method and to a third method of using Template Point Profiles and Template Point Centerlines, where a template ID "follows" a particular centerline and profile. One advantage of the Template Series approach is that it can be used to link different templates together, like non-curb and curb templates, as shown here in plan view:



For the above example, Template 1 applies from station 0+00 to 0+30, then transitions to Template 2 at 1+00, which itself transitions to Template 3 (still no right-side curb), which ends at 1+05. There, Template 4 starts with a curb replacing a standard EOP/Ditch combination on the right side. So Template 4 would be set to begin at 1+05.1, a short distance past 1+05. This template transitions into Template 5 at station 2+05. You do not need to enter start and ending templates at station 0+00 or after station 2+05. Therefore, the dialog for this example might look as follows:



Note that you can run Process Road Design to review the design results in plan view, with entry of only the Design Template/Series, the Profile and the Centerline (items 1, 2 and 4 within Process Road Design). You do not need existing cross sections to use Process Road Design. If you process at an interval such as 10 over any desired station range, you can output the Template Polylines and verify the result in plan view. If no sections are found, the program will process from edge of shoulder left to edge of shoulder right, and omit cut and fill slopes. With the correct templates, this would reproduce the plan view shown above.

Input-Edit Template Series is also an effective way to accomplish superelevation, and even simultaneous superelevation and lane widening. Consider the "stages" of pivoting into superelevation of 3%. The first template might be called "Normal Crown" (the lower template). The second template might be called "Reverse Crown" (+2% cross slope). The third template might be called "Full Super" and would be the +3% template. You need the second template because you need to "restrain" the left-hand side of the road from pivoting until the continuous +2% cross slope is reached. If you only used the "Normal Crown" template, say, at station 4+00 and then the "Full Super" template at station 6+00, then at station 5+00, where 1/2 of the transition occurs, the left side cross slope would be -2.5% (transitioning halfway). In reality, the left side should not pivot until station 5+60. If the rate of pivoting is less from normal crown to flat outside lane, and the rate changes after that point, then you would need a fourth template to direct how the road transitions to full superelevation.

Prompts

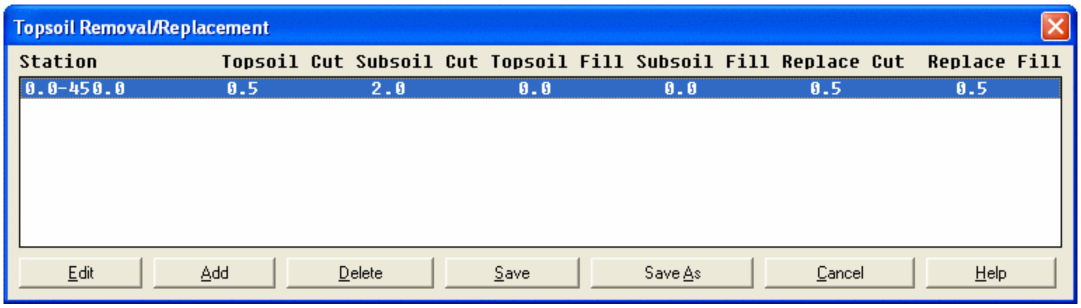
Keyboard Command: tplseries
Prerequisite: Template Files
File Name: \lsp\tplmake.arx

Topsoil Removal/Replacement

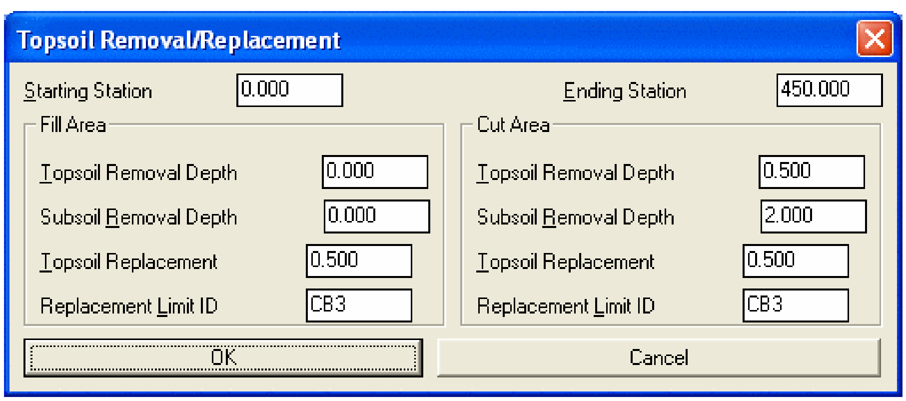
This command creates a topsoil definition (.TOP) file which defines topsoil removal and replacement zones to be used in the *Process Road Design* command. You can have different topsoil adjustments for different station ranges. These adjustments are applied to the existing ground section in the *Process Road Design* command and will effect the cut and fill volumes. *Process Road Design* will also report the amounts of topsoil removal and replacement.

The command starts by displaying a list of the topsoil stations in the dialog shown below. To add a topsoil adjustment, pick the Add button which brings up a second dialog. You can have different amounts of topsoil removal and replacement for areas in cut and areas in fill. Subsoil is another category of removal that will be combined

with any topsoil removal. The Subsoil removal volume is reported separately from topsoil removal by Process Road Design. Subsoil is automatically removed from the site and not used in fill or as a replacement quantity. Therefore, the subsoil element applies only to unsuitable materials that need to be removed. In the example below, we are only removing topsoil in cut (where cutting must take place in any case), and in the cut, we are removing 2' of subsoil which will be hauled off site (since subsoil is not re-used). The removed 0.5' of topsoil in cut will then be replaced in both cut and fill zones of the road within the limits specified by the "Replacement Limit ID". (No topsoil will be replaced on paved surfaces!)



The Replacement Limit ID is an option to limit the replacement to occur only within the template left offset Limit ID and the right offset Limit ID. If this Limit ID is left blank, then the program will apply the replacement between the left catch point and the right catch point. Topsoil removal is always applied between the catch points. The Limit ID corresponds to a template ID as set in the Design Template routine. Typically, you would use an ID like SH for shoulder and replace topsoil only from the far left and right tie/catch points to the SH or shoulder point. If you use a curb and want to replace topsoil to back of curb, keep in mind that the program takes the basic code "CB" and creates 3 curb points typically, so the back of curb would become CB3 in most L-shaped curbs.



If the Topsoil (".TOP") file is selected within Process Road Design, all quantities of topsoil removal and replacement and subsoil removal are reported, as shown below:

Processing 0+00.00 to 4+42.10

Total Topsoil Removed: 5219.22 C.F., 193.30 C.Y.
Total Subsoil Removed: 20876.89 C.F., 773.22 C.Y.
Total Topsoil Replaced: 5309.57 C.F., 196.65 C.Y.
Hauled-In Topsoil: 90.35 C.F., 3.35 C.Y.

Total Cut : 9106.52 C.F., 337.28 C.Y.
Total Fill: 16402.56 C.F., 607.50 C.Y.
Total SUBGRADE1 - asphalt: 2763.36 C.F., 102.35 C.Y.
Total SUBGRADE2 - stone: 9209.44 C.F., 341.09 C.Y.
Total CURB - concrete: 1078.37 C.F., 39.94 C.Y.

The cut reported in Process Road Design would be the remaining cut after topsoil and subsoil removal, and the fill would be the fill necessary to bring the grade to base of topsoil replacement, on top of which the topsoil is added. The removal of topsoil and subsoil usually creates less cut and more fill, as some of the cut is accomplished by the topsoil/subsoil removal, and in terms of fill, the grade must be brought up to replace the "cavity" created by the topsoil and subsoil removal. Topsoil removal depths and replacement depths can have a dramatic impact on cut and fill quantities, particularly on smaller scale projects like subdivision roads. In this example, every extra 0.1' of topsoil removal produces approximately 100 c.y. of net fill.

Prompts

Topsoil File to Read Specify a topsoil file.
Topsoil dialog Choose your options.

Keyboard Command: topsoil
Prerequisite: None
File Name: \lsp\eworks.arx

Assign Template Point Profile

This command assigns profile (.PRO) files to template point ID's like EP (edge of pavement), SH (shoulder) or DL (ditch line), storing this information in a template point profile (.TPP) file which can be used by the *Process Road Design* and *Road Network* commands. The purpose of the profile assignments is to allow separate profiles for template points that are independent of the centerline profile. For example, a ditch grade could have a different profile than the centerline. Multiple template point profiles can be assigned so the amount of control is unlimited. The Template Point Description corresponds to the name set in the *Design Template* command.

If you want the template ID point to follow a special slope or vertical alignment, use Assign Template Point Profile. The combination of using template point centerlines and profiles applied to particular template ID points is a design method sometimes referred to as "strings", where template elements string along special horizontal and vertical alignments. The rules of the template in terms of distances and slopes to the next point in the template will resume after the template point centerline and profiles are applied.

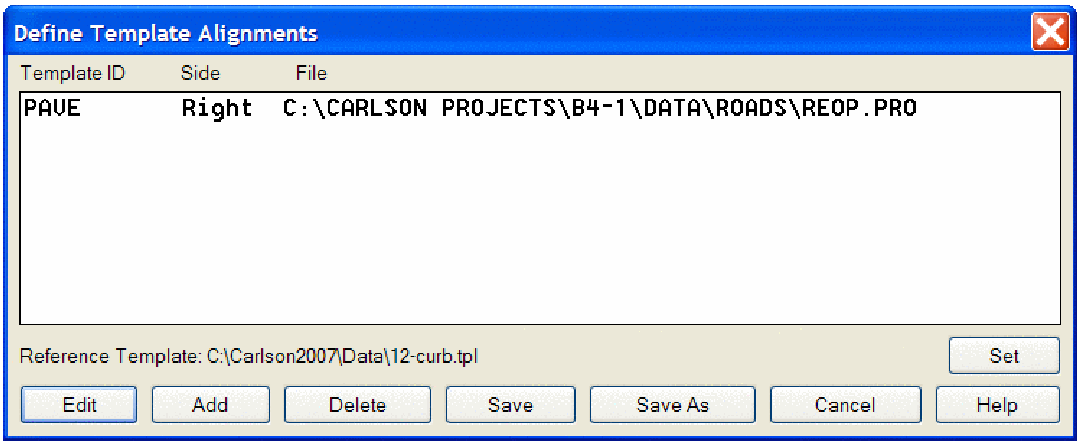
Prompts

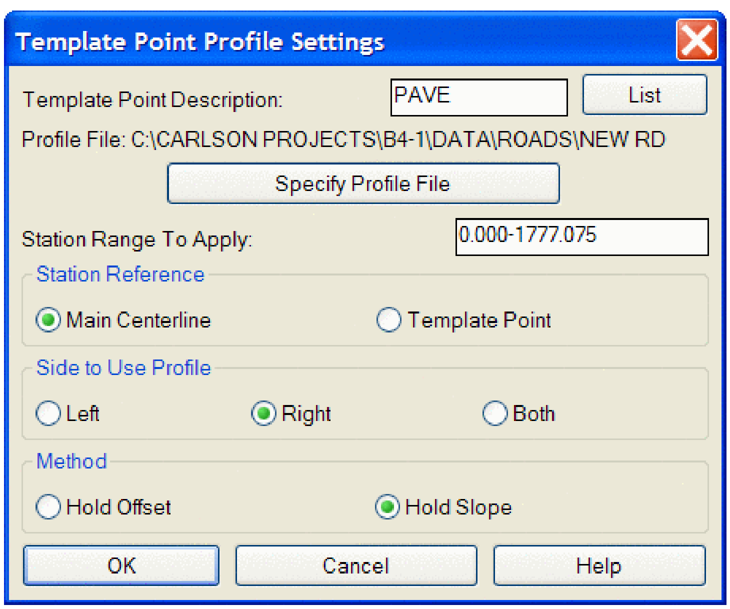
First you are prompted to create a new Template Point Profile (.TPP), or edit an existing one.

Next the Define Template Alignments dialog is presented, showing a list of exisitng Template ID-Profile assignments. To add a new assignment, first pick the Set button to set the Reference Template file (.TPL), then pick the Add button. This brings up the Template Point Profile Settings dialog. First, pick a Template Point Description from the List, which is derived from the components defined in the Template. Next, pick the Specify Profile File button, to choose the file (.PRO) to assign to the Template Point ID. Enter the Station range to Apply the assignment, select the Station Reference, specify if this assignment is for the Left, Right, or both sides of the main centerline, and

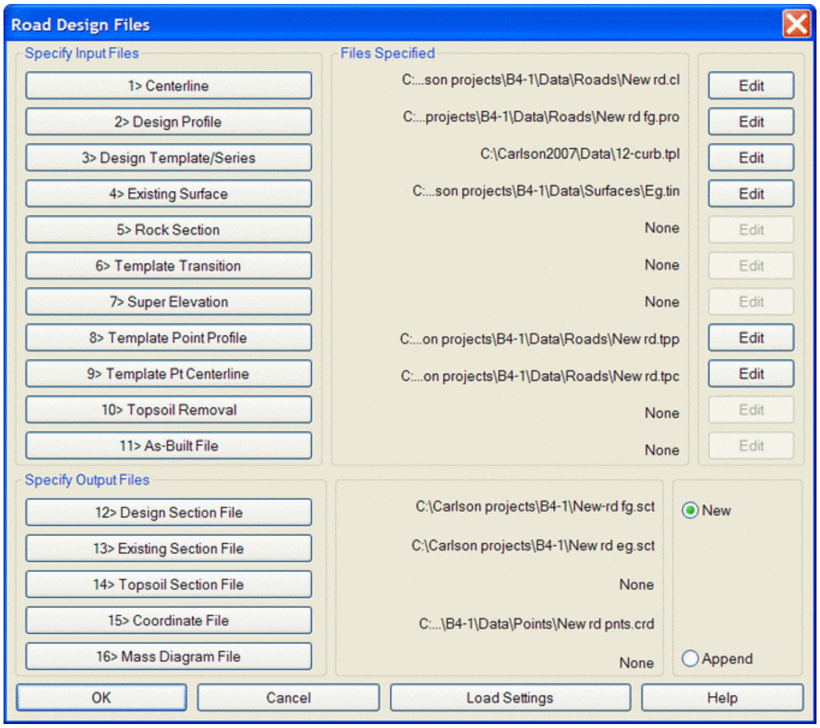
finally specify the method to apply the assignment. Since the template ID profile can change the relative position of the template ID from the centerline, you have two options for how to fit in the template ID profile: Hold Offset or Hold Slope. Hold Offset will keep the same offset for the template ID and adjust the slope to the template ID. The Hold Slope will keep the same slope to the template ID and adjust the offset to reach the template ID profile elevation. Use Hold Offset when Template Point Profile is used in conjunction with Template Point Centerline, where a single template ID is defined to follow both a special and distinct horizontal alignment (centerline) and vertical alignment (profile).

Pick OK. Back in the Define Template Alignments dialog, pick Add to add another assignment, Edit to edit an existing assignment, Delete to delete a defined assignment, or Save to Exit.

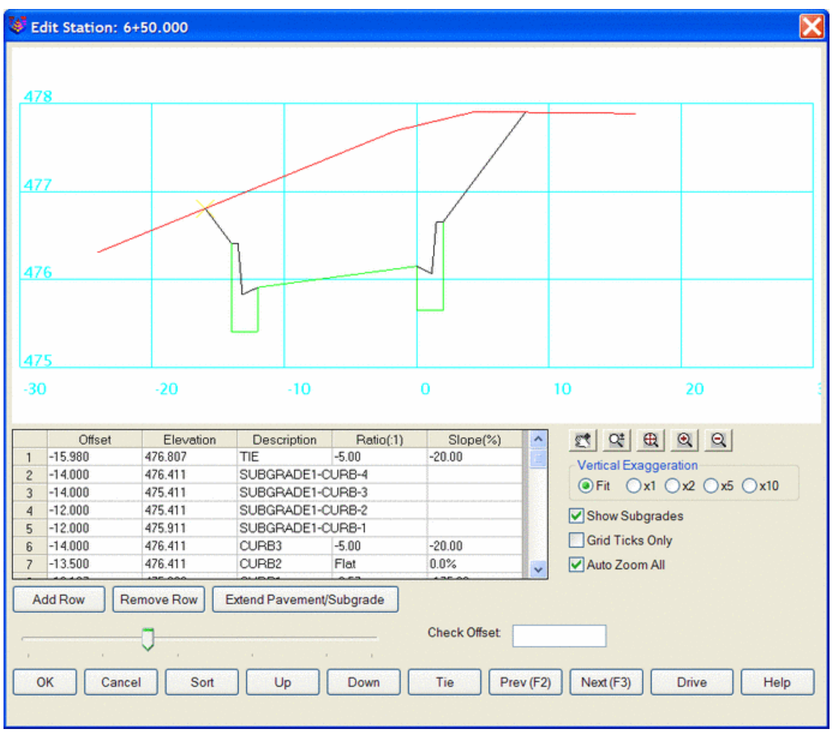




Now Process the road design employing the newly defined Template Point Profile assignment. This is the Process Road Design main dialog. Pick the Template Point Profile button to select the new file (.TPP). You could also create a new Template Point Profile file directly from this dialog box.



Viewing the road sections with the *Input-Edit Section File* command on the Section menu shows the effect of the Template Point Profile assignment.



Keyboard Command: tppset
Prerequisite: Profile file (.PRO)

File Name: \lsp\eworks.arx

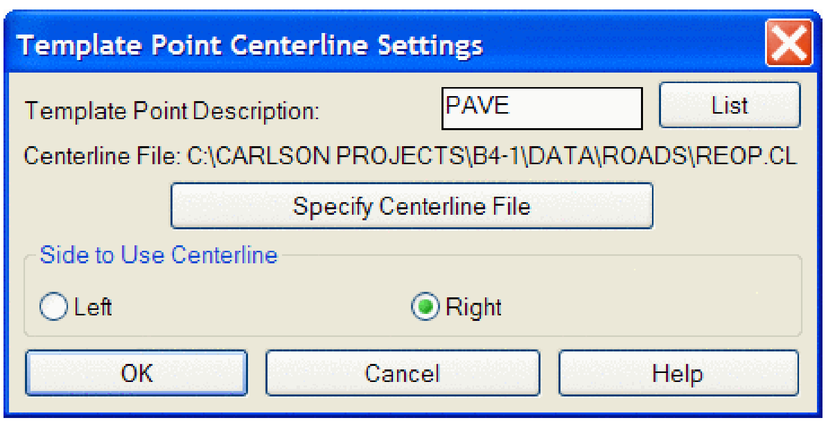
Assign Template Point Centerline

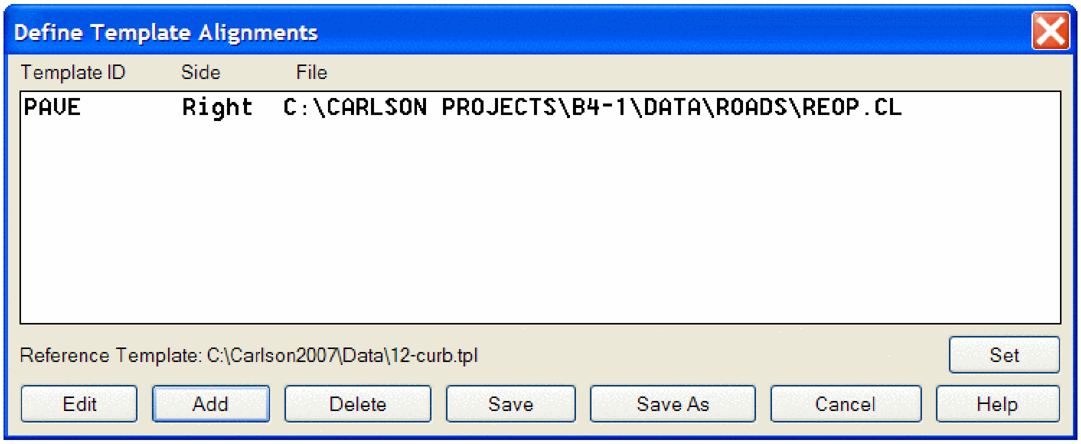
In roadway design situations involving varying pavement widths, the only effective way to control the edge of pavement positions is through the use of Assign Template Point Centerline. This command assigns centerline (.CL) files to template ID points, independent of the main centerline, thereby controlling the horizontal location of the edge of pavement. The assignment of Template ID points to centerline files (.CL) is stored in Template Point Centerline files (.TPC). These files are then used by the *Process Road Design* and *Road Network* commands. The slope to these template points is based on the parameters defined in *Design Template*. Subgrades can be made to follow template IDs if their offset distances are defined not by distance but by reference to the template ID.

Prompts

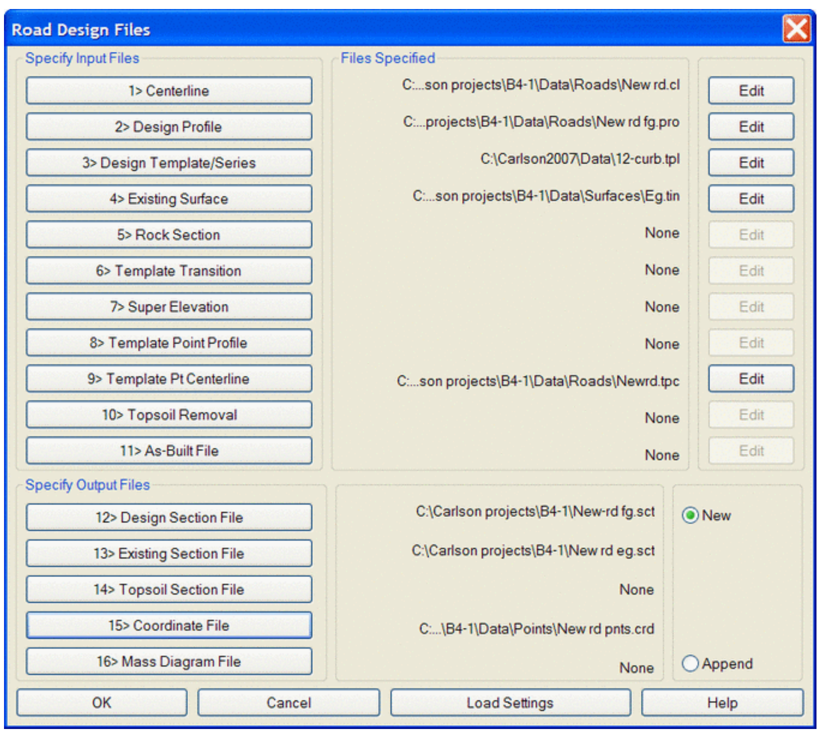
First you are prompted to create a new Template Point Centerline file (.TPC), or edit an existing one.

Next the Define Template Alignments dialog is presented, showing a list of existing Template ID-Centerline assignments. To add a new assignment, first pick the Set button to set the Reference Template file (.TPL), then pick the Add button. This brings up the Template Point Centerline Settings dialog. First, pick a Template Point Description from the List, which is derived from the components defined in the Template. Next, pick the Specify Centerline File button, to choose the file (.CL) to assign to the Template Point ID. Finally, specify if this assignment is for the Left or Right side of the main centerline. Pick OK. Back in the Define Template Alignments dialog, pick Add to add another assignment, Edit to edit an existing assignment, Delete to delete a defined assignment, or Save to Exit.



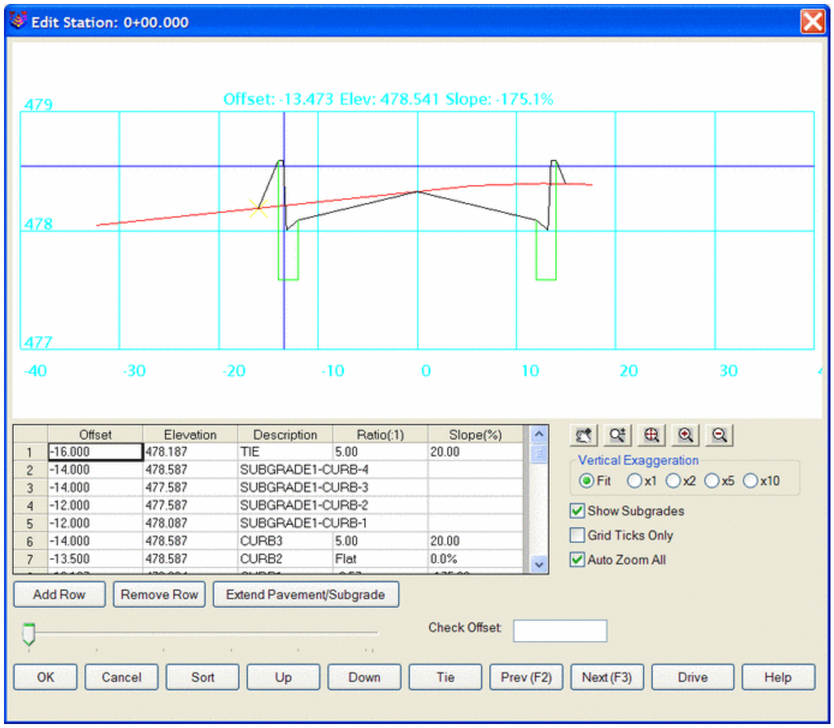


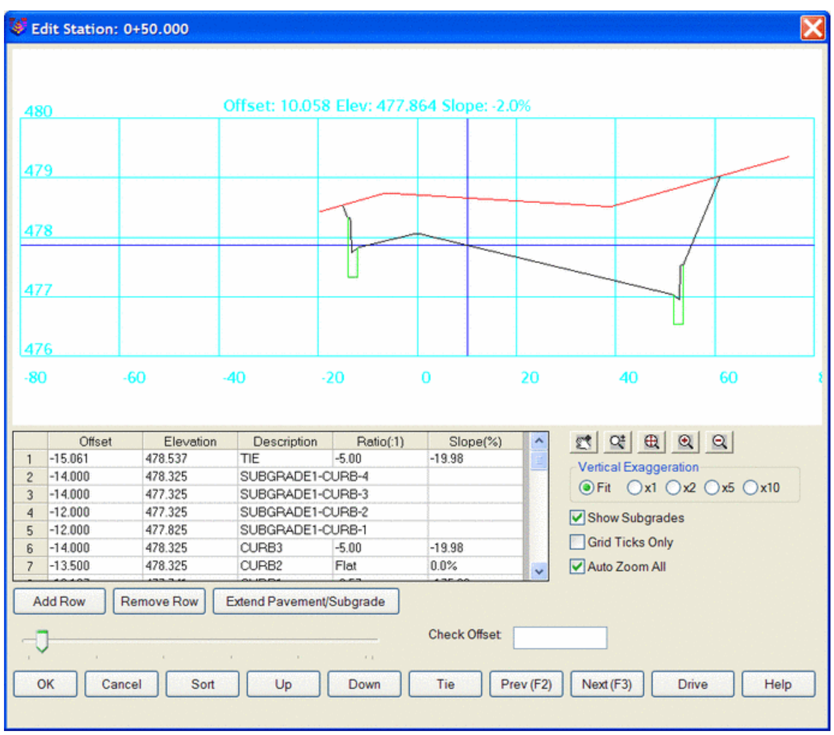
Now Process the road design employing the newly defined Template Point Centerline assignment. This is the Process Road Design main dialog. Pick the Template Pt Centerline button to select the new file (.TPC). You could also create a new Template Point Centerline file directly from this dialog box.



Pick OK to Process the road design.

Here are two sections along the roadway, illustrating the varying lane widths on the right side of the main centerline. They are viewed with the *Input-Edit Section File* command on the Section menu.



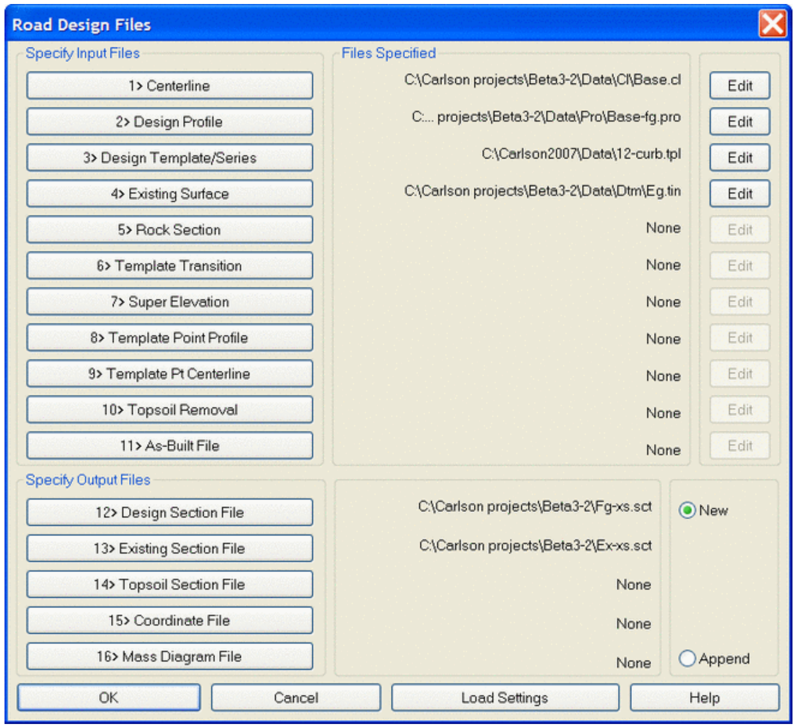


Keyboard Command: tpcset
Prerequisite: Template file (.TPL), secondary Centerline file(s) (.CL)
File Name: \lsp\eworks.arx

Process Road Design

The primary function of this command is to assemble all of the components for a road design and process them together. While all of the Input Files can be created prior to accessing the Process Road Design command, all can be edited from the Road Design Files dialog, and many files can actually be created from the Road Design Files dialog itself. The actual processing of the Road Design essentially applys the design template at the design profile elevation along the specified centerline and computing the outslopes and earthworks relative to the existing ground surface. The earthworks report can be shown in the standard report viewer or customized with the Report Formatter option. Secondary functions include creating a final grade section file for plotting with the *Draw Section File* command, creating final grade points in a coordinate file, creating a final surface/contour model, and drawing the road as 3D polylines. You can also output a mass haul diagram profile. The program also has options for applying a superelevation file, template transition file, template point profile, template point centerline, rock section file, an as-built existing section file and a topsoil removal file. *Process Road Design* can be used not just for final road design computations but for levees, channels and any template-based application.

This command begins with the dialog shown below. The top section contains input Files. In a typical implementation of this command, you will have already defined a horizontal centerline for the design to follow, however, you could actually pick the Centerline button, pick the New tab, name the new centerline file (.CL), pick Open, and then back in the main Road Design Files dialog, pick the Edit button and layout the centerline design. The only component that you must have already created before running Process Road Design is #4, an Existing Surface file. As long as there is an Existing Ground Surface, the command will generate the Existing Ground Profile automatically, and the Proposed Finish Grade Profile can be created with the Edit button. Even a Design Template can be created right from here as well. Ultimately, the top 3 Input items (Centerline, Design Profile, and Design Template/Series) are required to Process a Road Design, leading to final sections and full contouring and 3D viewing. The Existing Surface is needed as well to process with earthwork calculations and tie slopes.



Input items 5 through 11 are strictly optional design files. It should be pointed out that items 8 and 9 (Template Point Profile and Template Point Centerline) enable template IDs to follow any defined centerline or profile and provide total flexibility of design. Lane widening, matching existing curb lines, special ditches, etc. can be easily accomplished with these two options. The template IDs simply "string along" or follow these pre-defined alignments, and the rules of the template apply to all other template ID points.

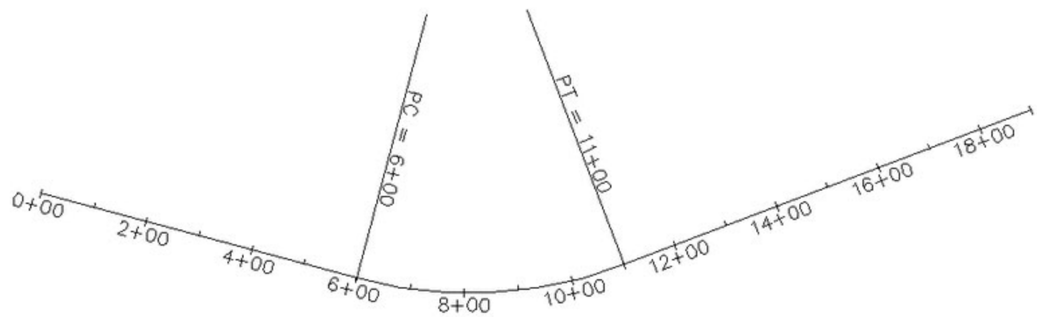
The Output Files section allows you to specify files to store the processing results. The Section File creates a final grade section file that can be drawn with *Draw Section File.* The Topsoil Section File creates the modified existing ground section file if Topsoil Removal is set in the input. This "post-topsoil removal" section file can be used for earthworks calculations to compare any stage of work, using *Calculate Sections Volume* under the Section pulldown menu. The Coordinate File creates a coordinate file containing every break point in the final grade. The point descriptions include the station, offset and template ID. Whether to include the subgrade points as well as the final surface points is determined by the Include SubGrade Points in Output CRD File option on the next dialog. To the right of the Output Files is the option to create new output files or append to existing output files. If you extend the road, or revise a portion of the project, you can simply "Append" rather than overwrite. The first time that you run this command for stations 0-1000, you would set Output Files to New. Then you could run this command again, possibly with new inputs, for stations 1000-2000 and set Output Files to Append.

On the next dialog, there is a Save Settings button to store all the settings from the first and second dialogs into a specified Road Design File with an (.RDF) file extension. Recorded (.RDF) files can be recalled later using the Load Settings option.

1> Centerline

Specify the name of the Centerline file with this option. The (.CL) file contains the horizontal alignment geometry for a project. This parameter file must be specified if you want to have earthworks centroid corrections computed, generate final coordinates, Disturbed Area Polyline, and/or use Triangulate & Contour. The centerline file can be

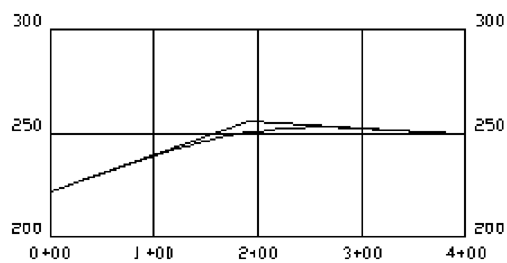
created by the *Design Centerline* or *Polyline to Centerline* commands in the Design pulldown menu.



Example Centerline

2> Design Profile

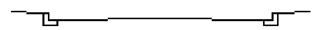
Specify the design profile (.PRO) file to derive the centerline elevations when the template is applied. This file defines the vertical alignment and is always required. The profile can be created with any of the profile creation routines in the Profile menu, but typically you would use *Design Road Profile* or *Input Edit Profile.*



Example Design Profile

3> Design Template/Series

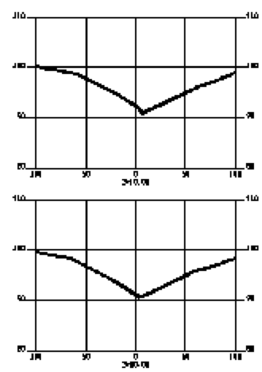
Specify a template definition (.TPL) file or template series (.TSF) file that defines the final grade offsets and elevations and the cut/fill slopes. The template file is created by the *Design Template* command and the template series file (a set of templates ordered by range of stations) is created using *Input-Edit Template Series*. A single template file or a template series file is required to run *Process Road Design.*



Example Design Template

4> Existing Surface

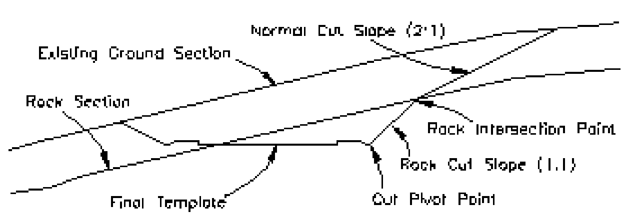
Specify the surface model which will be treated as the existing ground for cut and fill volumes and to calculate the outslope intersections when the template is applied at the profile elevations. This Existing Surface can be defined by either a section file or triangulation. The section file can be created with commands such as *Sections from Surface Entities*, *Input/Edit Section File*, *Sections from Points* or one of the *Digitize Sections* commands on the Section menu. The triangulation file can be created with the *Triangulate & Contour* command.



Example Existing Sections

5 ≫ Rock Section File

This option specifies an optional rock section file that is used as an additional surface. When in cut, a special cut slope is used up to the intersection of the rock surface. After this intersection, the normal cut slopes apply. The special rock cut slope is specified in Design Template under the cut options. If the "pivot point" in cut is below the rock line, then the special rock cut slope will be applied. Note that rock sections can be derived from borings to rock, as modeled, or can be created quickly by using the "translate" command within *Input-Edit Section File* to translate the existing ground sections by a vertical offset (e.g. -6) to an approximate top of rock.



Detail of rock cut slope

6 ≫ Template Transition File

Specify a .TPT file with this option. The Template Transition file allows modified template files to be applied at different ranges of stations on a project. In this way, template IDs can be made to widen (as for passing lanes) and contract. Use the *Template Transition* command under the Design menu to create a template transition file.

7 ≫ Super Elevation File

This option is used to specify a super elevation file (.sup file) that defines the super elevation transition stations on a project. The super elevation file can be created with the *Input-Edit Super Elevation* command.

8 ≫ Template Point Profile

This option lets you have separate profiles for template points that are independent of the centerline profile. This design file is created with the *Assign Template Point Profile* command.

9 ≫ Template Point Centerline

This option lets you have separate centerlines for template points that are independent of the main centerline. This design file is created with the *Assign Template Point Centerline* command.

10 ≫ Topsoil Removal

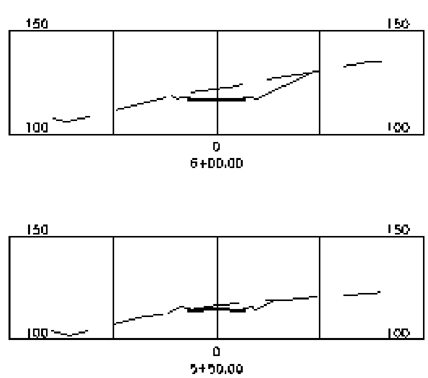
This option applies topsoil removal and/or replacement to the existing ground section file. This design file is create with the *Topsoil Removal/Replacement* command.

11 ≫ As-Built File

The As-Built File is a cross section file used to match existing grade and retain as-built portions of a road improvement project. The final cross sections will conform to the as-built cross sections for those template IDs specified in the second dialog. Beyond the specified set of offsets in the as-built cross section file, the design road files will be applied.

12 ≫ Output Design Section File

Specify the name of the file to output the final grade sections calculated by applying the template file at profile elevations and calculating the outslope intersection with the existing ground cross sections. This file can then be plotted by using the *Draw Section File* command. After plotting the final sections overlaid on the existing sections, revisions can be made graphically with commands like *PEDIT* and *Polyline by Slope Ratio*. The data output to the file can also be edited and reviewed with the *Input-Edit Section File* command. If the final sections are edited graphically, the revised section data can be updated in the .SCT file with the *Polyline to Section File* command.



Output Section File drawn with Existing Section File by the *Draw Section File* command

13 ≫ Output Existing Section File

This option creates a section file of existing ground. This applies when the existing surface is a triangulation file. The station intervals for the existing section file will match the stations from the design section file.

14 ≫ Output Topsoil Section File

This option writes out a modified existing ground section adjusted by the topsoil removal. This option is only valid if a Topsoil Removal file is being used.

15 ≫ Output Coordinate File

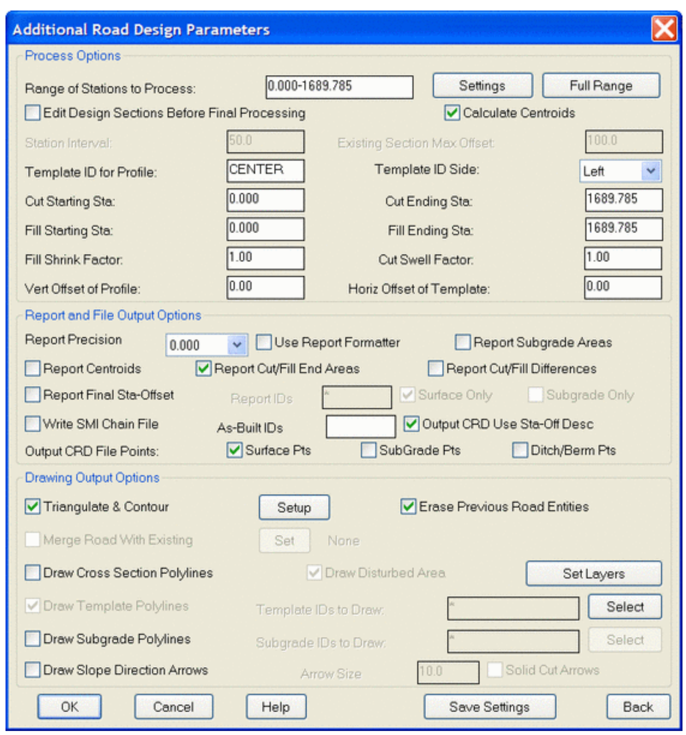
This option creates a coordinate file containing every break point in the final grade for the range of processed stations. Using the second dialog, there are additional options to output subgrade and ditch/berm points. The point descriptions include the station, offset and template ID. The station interval is set by the stations in the Existing Section File.

16 ≫ Output Mass Diagram File

The mass haul diagram can be output as a profile file and shows the cumulative cut and fill along the selected range of stations. Cut and fill is balanced between points on the mass haul profile that cross the Z-axis. Because of the typically large values of cut and fill associated with road and earthwork projects, the vertical scale for the profile may need to be set to 10 times the horizontal scale, or more. The profile preview screen which appears when you select profile for loading will show the elevation range and help suggest an appropriate vertical scale.

Running the Road Design Job

After setting up the files and options in the first dialog click the OK button. The next dialog shown below has processing options.



In the **Process Options** section, the *Range of Stations to Process* field sets the range of station that you want to calculate. Each time you use this command, the existing grade (.SCT) file is scanned and the range in the edit box

is set to the minimum and maximum stations in the file. If you change the station range, you can click the *Full Range* button to restore the default full range of stations. The *Settings* button will interpolate additional existing cross sections (internally) and create final cross sections at special stations like profile high and low points, profile transition stations for PVC and PVT, key centerline points like PC's and PT's, and superelevation and template transition points and any user-defined special stations. These additional station improve volume calculations.

The *Edit Design Sections Before Final Processing* does just that. You can review and edit the final sections in the spreadsheet with graphic view editor similar to the Input-Edit Section File command. For example, you can change the tie slope as selected stations. After making these changes, the modified final sections are used for the rest of the road design process including earthworks and drawing output.

The *Station Interval* and *Existing Section Max Offset* buttons are ghosted if the existing surface is a set of cross sections. If there is no existing surface, or the existing surface is a grid, TIN or FLT file, then you must enter the Station Interval to generate sections along the centerline. Besides the stations at interval, sections can be created at special stations as specified under the Settings button. The *Existing Section Max Offset* controls the max left and right offsets for generating the existing sections when the Existing Surface is defined by a triangulation file. This offset needs to be set far enough for the final sections outslopes to tie into existing. On the other hand, keeping this offset fairly close to the tie point will help make processing run faster.

The *Calculate Centroid* option applies to centerlines containing curves. The centroids of the cuts and fills will be computed, and the radius to these centroids will be calculated. Then the effective interval will be computed between cut and fill centroids. In this way, in a tight curve where fill is concentrated to the outside of the curve and cut is concentrated to the inside of the curve, fill will be increased and cut will be reduced. This also increases the accuracy of volume calculations.

The *Template ID for Profile* allows the profile grade to be applied to another template ID point other than the centerline. This feature might apply, for example, to a 2-lane road that will eventually be part of a 4-lane road being built in stages. The first-stage, 2-lane road would be fully symmetrical and designed around the crown of the road, but the template profile might be one of the edge of pavements. You can specify the template ID (e.g. EP), and whether the left or right side ID should be used to apply the profile grade.

Volumes are calculated using end areas between the range of stations. Instead of cutting off the volumes exactly at this range, the Ending and Starting Stations for Cut and Fill can be used to have the volume taper from zero at the specified Starting Station to the volume at the first station in the range. Likewise the Ending Stations can be used to taper the volume from the last station in the range to zero at the specified Ending Station.

The *Shrink* and *Swell Factor* edit boxes allow you to specify a value that the volume calculated will be multiplied by. If you specify any number other than one an additional report showing accumulated adjusted volumes and differences will be produced.

The *Vertical Offset of Profile* edit box will place the template at the profile grade as raised or lowered by the entered offset. The *Horizontal Offset of Template* will shift the template left or right on the centerline by the specified amount. Use a positive value to offset to the right and use a negative value to offset left. This option is useful, for example, when one side of a divided highway is built years before the other side is to be started. In this case, you could define a normal template with a crown in the middle, but would enter a horizontal offset from the crown of the road to the actual centerline of the divided highway.

The **Report and File Output Options** include settings for reporting final coordinates (if specified in the previous file output dialog), as well as special features.

The *Report Precision* controls the number of decimal places.

The *Use Report Formatter* option allows you to customize the fields to report and their order. It also can output the report to MS Excel or databases.

The *Report Subgrade Areas* option will include an additional line in the report for the end area of each subgrade material.

The *Report Centroids* toggle controls whether the shift in the cut or fill centroid radius shift will be included in the earthworks report.

The *Report Cut/Fill Text* option greatly expands the size of the report by presenting the cut and fill end areas at each station. A sample of the cut/fill text report is shown below. Volumes by end area method are presented between each line containing station and end areas of cut, fill and optionally rock.

```
 Station   Cut(sf)  Fill(sf) Rock(sf) Interval  Cut(cy)  Fill(cy)  Rock(cy)
3+00.00   0.00     101.07   0.00
                                     50.00     313.78    93.58     0.00
3+50.00   338.88   0.00     0.00
                                      6.09      80.93     0.00     0.00
3+56.09   379.10   0.00     0.03
                                     43.91     824.60     0.00    31.84
4+00.00   634.92   0.00    39.12
```

The *Report Cut/Fill Differences* option will report the cut/fill ratio and balance at each station.

The *Report Final Station-Offset* option will create a report of the final section offset-elevation data in row-column format. The station and profile grade are shown on the left followed by columns of offset and elevation for each data point. There are options to report the surface points only, the subgrade points only or filter the points by ID.

Write SMI Chain File creates a chain (.CH) file that contains the centerline, profile and template data for SMI Construction V.

The *As-Built IDs to Use* option applies only if you have specified an as-built section file as one of the inputs in the previous dialog. Consider a normal road template with 20 feet to edge of pavement (EP) and 10 feet more to shoulder (SH). Going further, assume that when you run this template, it does a fill condition on the right and creates a TIE point. If you wanted to conform the template to match a wider section of road at certain stations, you could edit the output file of a normal run (using Input-Edit Section File) and create new offsets and subgrade points for widening and even force a trapezoidal ditch in cut, as shown in the entries below:

	Offset	Elevation	Description	Ratio(:1)	Slope(%)
12	21.330	1996.950	EP	-50.00	-2.00
13	21.330	1995.920	SUBGRADE1-3		
14	31.500	1996.544	SH	-25.00	-4.00
15	21.330	1996.950	SUBGRADE1-4		
16	38.600	1992.994	BD	-2.00	-50.00
17	40.000	1992.994	BD2	Flat	0.0%
18	46.000	1995.994	TIE	2.00	50.00

Because all the other offsets to the left match by default, this editing will force the template to conform from offsets 21.33 right to the tie at 46 right. As you try different design template or other changes in *Process Road Design*, this as-built information would hold for the specified station. Alternately, you could edit the final cross section directly in

Input-Edit Section File. Note that you can use distinct, new ID points like BD2 which are not found in the template file, and they will be created if part of the as-built cross section file, and if referenced as *As-Built IDs to Use*. This As-Built method works best when inserting controlled section defined from TIE left to TIE right, which get inserted as completed sections in a run of Process Road Design.

The *Output CRD File* options apply when a Output Coordinate File is specified in the first dialog. These options allow you to output any combination of template surface, subgrade, ditch and berm points. The *Output CRD To Use Sta-Off Desc* option sets whether to include the station and offset in the description for each point. Here are example coordinates for station 0+90:

```
PtNo. North(y)  East(x)   Elev(z)  Description
122  189497.42 611730.32 90.01    TIE 0+90.00L53.65
123  189461.43 611733.72 108.09   SHD 0+90.00L17.50
124  189457.45 611734.09 107.93   CURB3 0+90.00L13.50
125  189456.95 611734.14 107.93   CURB2 0+90.00L13.00
126  189456.95 611734.14 107.09   CURB1 0+90.00L13.00
127  189455.96 611734.23 107.09   EP 0+90.00L12.00
128  189444.01 611735.36 107.33   CENTER 0+90.00R0.00
129  189432.06 611736.49 107.09   EP 0+90.00R12.00
130  189431.07 611736.58 107.09   CURB1 0+90.00R13.00
131  189431.07 611736.58 107.93   CURB2 0+90.00R13.00
132  189430.57 611736.63 107.93   CURB3 0+90.00R13.50
133  189426.59 611737.00 108.09   SHD 0+90.00R17.50
134  189412.18 611738.36 100.85   TIE 0+90.00R31.97
```

The **Drawing Output Options** bottom section of the Additional Earthworks Parameters dialog contains output options which are only available when a centerline file is specified.

The *Triangulate & Contour* option will automatically run this command after Process Road Design is done to create the final contours. Triangulate & Contour uses the template 3D polylines to model the final surface, and the disturbed area polyline is used as the inclusion perimeter for the contours. With Triangulate & Contour clicked on, the *Setup* button becomes active. Picking Setup brings up the Triangulate & Contour settings including the contour interval and whether to draw 3D Faces. Also under Setup, there are controls for the colors of the 3D Faces for each template break point. With Triangulate & Contour active, Draw Template Polylines and Draw Disturbed Area Polyline are automatically turned on. The *Merge Road With Existing* option combines the road design triangulation with the existing ground surface and stores the resulting triangulation in the file specified with the Set button. This option is available when the Existing Surface is a triangulation file and the Triangulate & Contour option is active.

The *Erase Previous Road Entities* option will erase any entities from the drawing that were created in a previous run of Process Road Design using the same design files. This option allows you to easily re-run Process Road Design and update the drawing entities after changing one of the road design files.

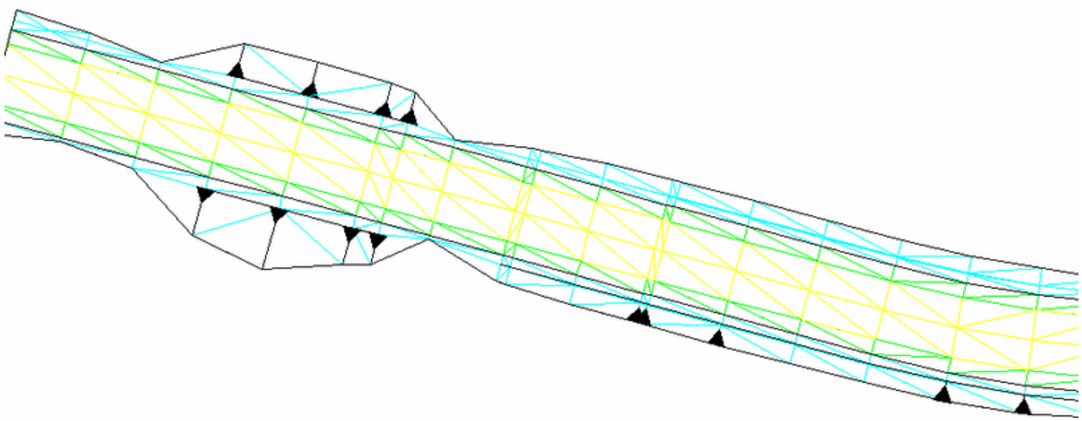
The *Draw Cross Section Polylines* option will create 3D polylines perpendicular to the centerline with each template break point. The interval of these cross section polylines is determined by the station interval of the Existing Sections.

The *Draw Template Polylines* option will create 3D polylines parallel to the centerline by connecting common template point IDs. For example, a template ID could be EP which this option would use to create 3D polylines for EP on the left and right of the centerline. Which template point IDs to connect in set under *Template IDs to Draw*. Setting this to an asterick (∗) will plot all the template break points. The *Select* button shows cross sections of the final templates for graphical selection of the ID's to draw.

Likewise, the *Draw Subgrade Polylines* option will create 3D polylines parallel to the centerline for the specified subgrade breakpoints.

The *Draw Disturbed Area Polyline* option will create a polyline perimeter that represents where the cut/fill slopes tie into the existing ground.

The *Draw Slope Direction Arrows* option will draw arrow indicators for cut or fill slope direction. The arrows are drawn in plan view and usually are drawn together with the Draw Disturbed Area and Draw Cross Section Plines options. Cut arrows start from the disturbed area limit and point towards the centerline. Fill arrows start from the base of the fill slope and point away from the centerline. The *Solid Cut Arrows* option chooses between solid fill or wire-frame cut arrows. These arrows, especially when drawn as solid cut arrows, help distinguish cut and fill at a glance, when in plan view. In the example below, fill from a berm is shown at the left and cut down to a ditch is shown at the right. The arrows will only draw if there is enough dimension in the cut and fill to fit the entire arrow. So the cut and fill arrows reveal the deeper cut and fill zones.



Prompts

Road Design Files dialog: Choose the design files
Additional Road Design Parameters
Road Design Report dialog
Trim existing contours inside disturbed area (Yes/<No>)? *Y* This prompt appears if Triangulate & Contour is on. This option will trim polylines with elevation that cross the disturbed area perimeter for the road.
Join final contours with existing (<Yes>/No)? *Y*This prompt appears if Triangulate & Contour is on. This option will join the final contours with the existing contours where they join at the disturbed area perimeter.

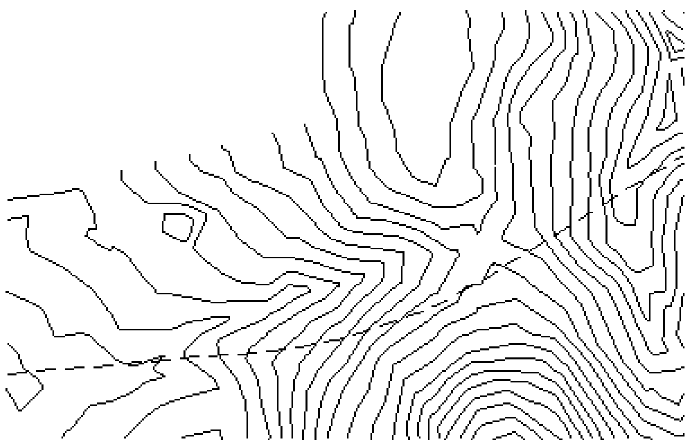
Portion of Earthworks Report:

```
 Template File> C:\DATA\simo2.tpl
Profile File> C:\DATA\rd.pro
Existing Section File> C:\DATA\simo2.sct
Centerline File> C:\DATA\simo2.cl

Processing 0+25.000 to 7+51.152
Total Cut : 800563.177 C.F., 29650.488 C.Y.
Total Fill: 1554948.266 C.F., 57590.677 C.Y.

Station  Cut(sf)   Fill(sf) Interval  Cut(cy)   Fill(cy)
```

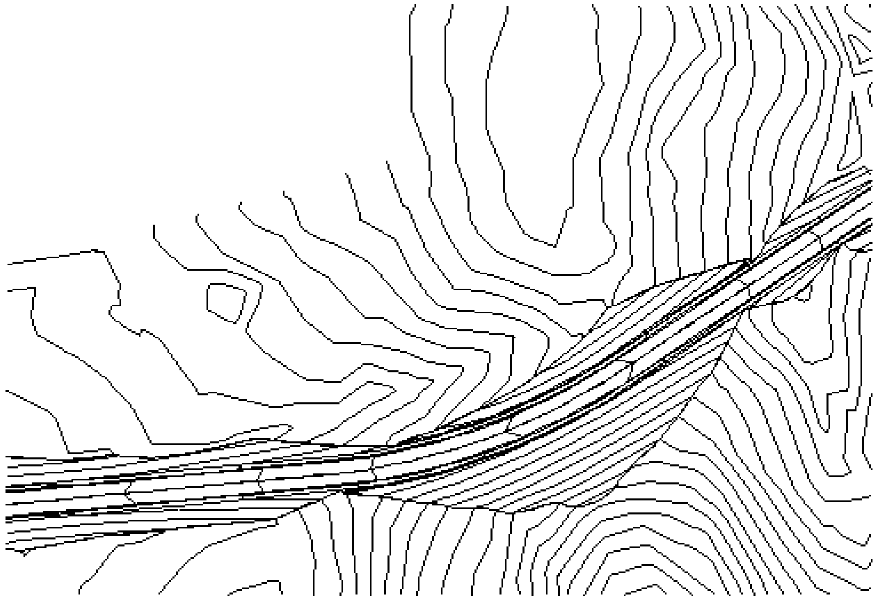
```
0+25.000 4407.456  0.000
                            25.000    4784.266  0.000

 0+50.000 5926.559  0.000
                            25.000    5535.921  0.000

 0+75.000 6031.029  0.000
                            25.000    4840.888  0.000

 1+00.000 4425.290  0.000
                            25.000    3432.528  0.000

 1+25.000 2988.971  0.000
                            25.000    2713.262  3.362

 1+50.000 2871.676  7.262
```

Portion of Final Station-Offset Report:

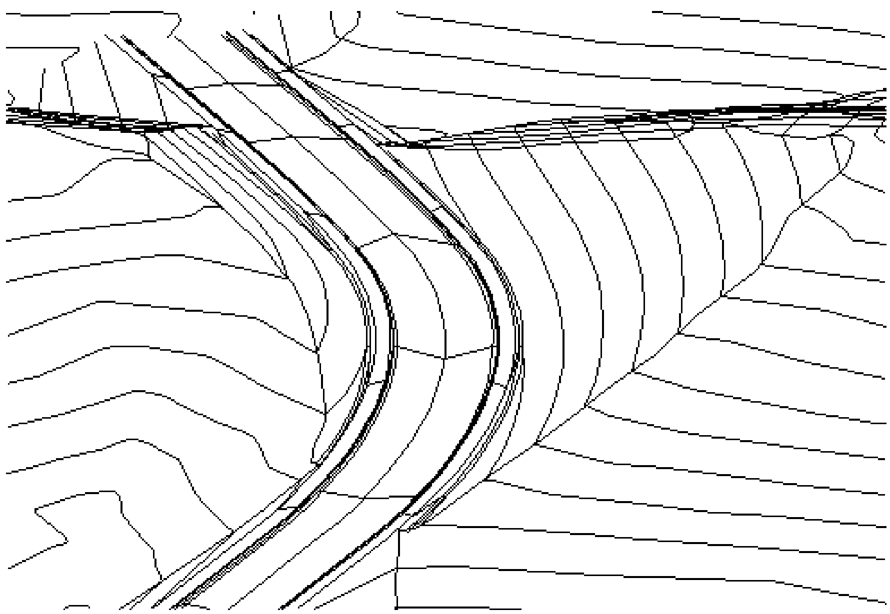
```
 Final Surface Section Report
STATION       P.G.
2+50.000      1013.444 59.619 18.000 12.000 0.000 12.000
              992.634 1013.444 1013.204 1013.444 1013.204
2+75.000      1015.059 65.772 18.000 12.000 0.000 12.000
              991.173 1015.059 1014.819 1015.059 1014.819
3+00.000      1016.499 71.547 18.000 12.000 0.000 12.000
              989.725 1016.499 1016.259 1016.499 1016.259
3+25.000      1017.764 76.733 18.000 12.000 0.000 12.000
              988.398 1017.764 1017.524 1017.764 1017.524
```



Existing Contours and Centerline



3D template polylines, disturbed area perimeter polyline and final contours



Template polylines and final contours viewed in 3D using Viewpoint 3D command

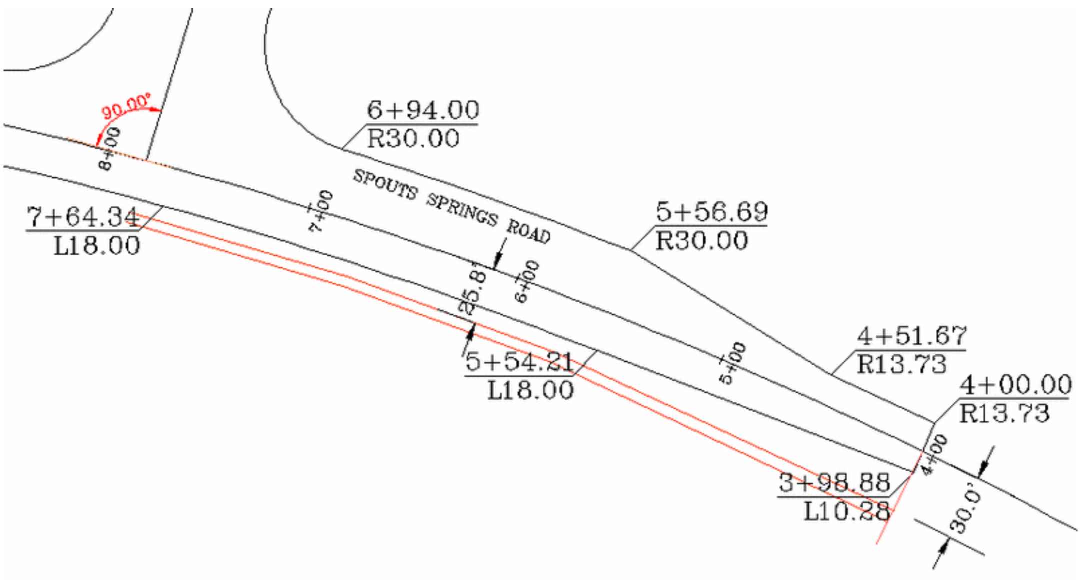
Review of 3 Methods of Transitioning Templates using Process Road Design

The 3 methods of template transitions and super elevation are:
(1) Template Transition and/or Super Elevation Files
(2) Template Point Profile and Template Point Centerline files
(3) Template Series file which transitions between multiple, named templates.

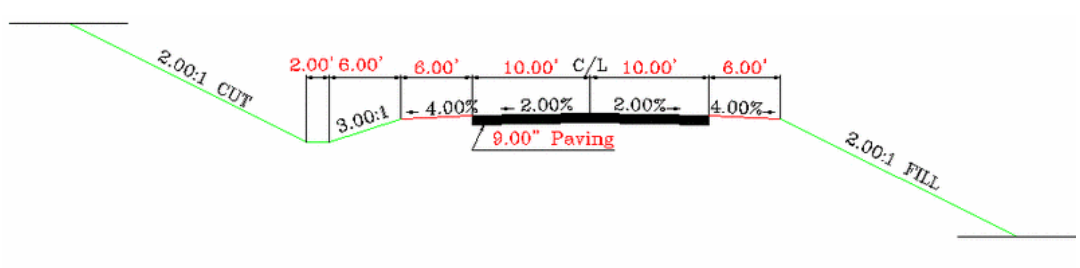
Road widening and lane transitions can be handled by all 3 methods. Special ditches are best handled by method (2), Template Point Profile and Template Point Centerline, especially since Template Transition files only work with

lanes or portions of roads defined by the Grade button in Design Template. Template Transition files do not apply to cut and fill segments, unless they are designed as fixed features using the Grade button. Super elevation can often be handled by method (1) or method (3). Bear in mind that new lanes or template elements that emerge and then disappear need to exist as template ID points in all referenced templates, using all 3 methods. These template ID points can be set to 0.001 units from adjacent template points, then "told" to emerge and widen as new lanes with distinct slopes appear. The program will not transition templates that don't share common template ID points.

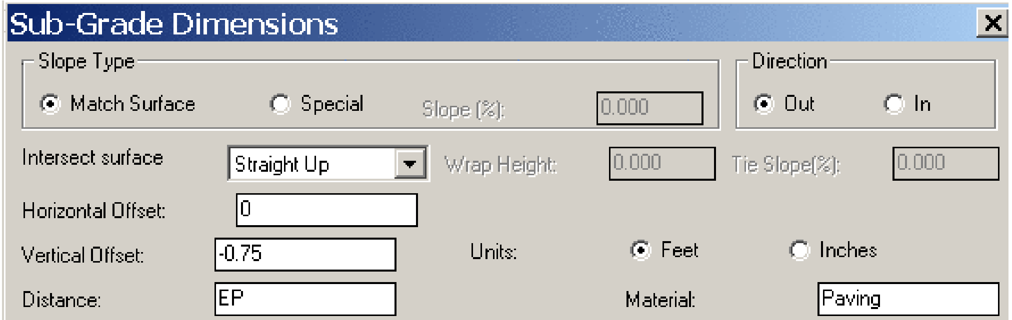
This deceptively easy looking example below might be approached by a combination of methods 1 and 2. For method 1 to apply (template transition), the slopes of the pavement lanes must be maintained according to the template definition from centerline to outside lane. The ditch portion will be handled by method 2 (template point centerline).



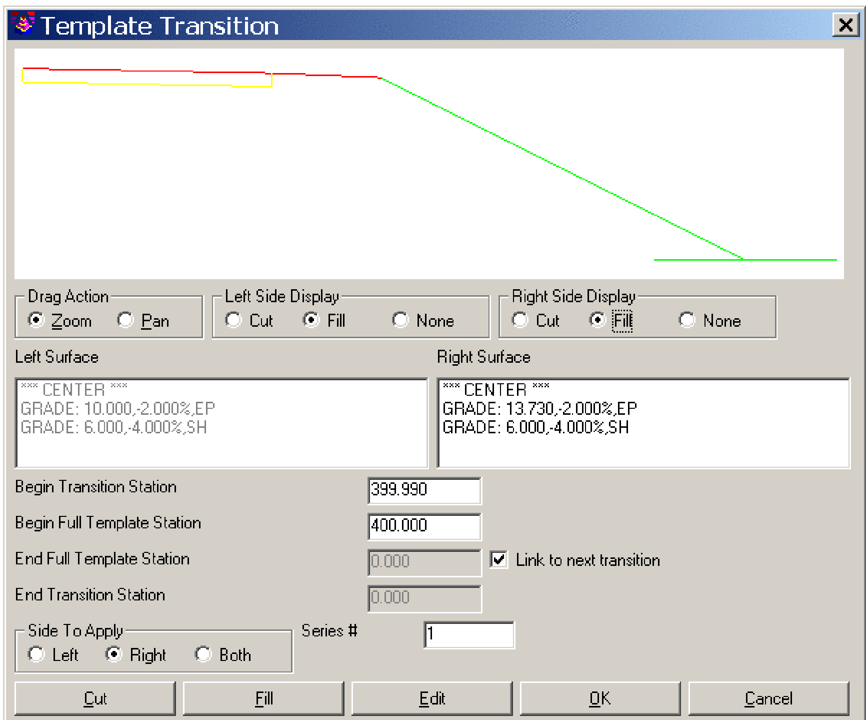
Assume Spouts Springs Road is a hillside road with a ditch cut on the left side and fill on the right side. The trapezoidal ditch is shown. We will design only from station 4+00 to station 6+94 where the intersection begins. The standard template of 10' left lane and 10' right lane might appear as shown below:



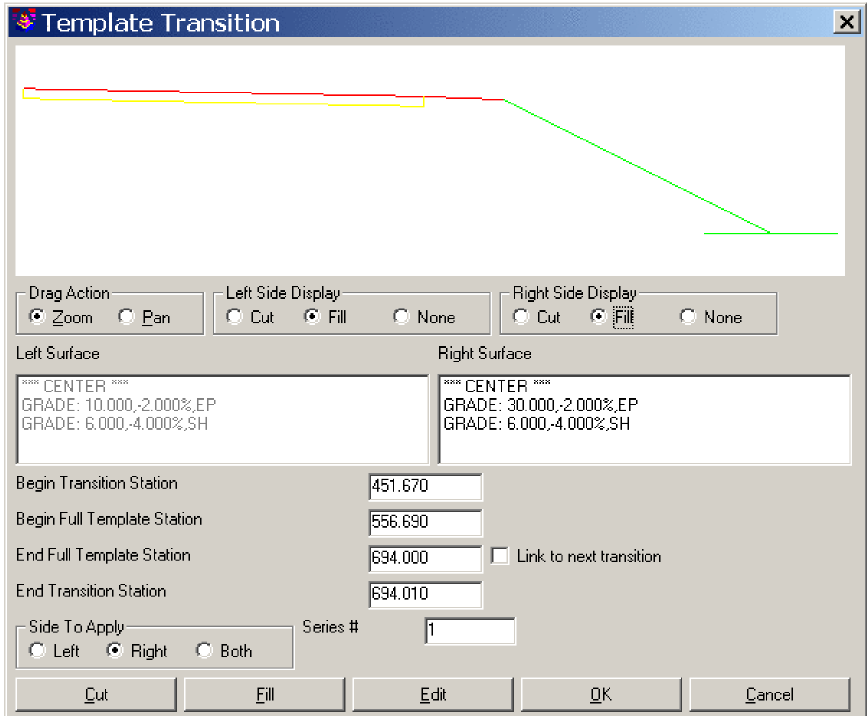
Note that if lanes are designed to expand, its important that the subgrade (9" of paving, shown above) be defined as following the ID, and should not be set to a fixed distance. The "EP" ID is used in the dialog below (top of subgrade dialog within Design Template) for this example.



The right hand portion of this example would be entered as follows:

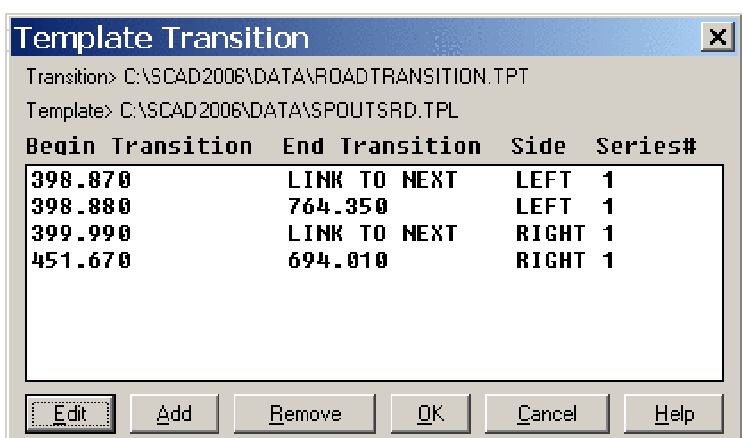


When you click "Add" within the Template Transition main dialog, you are presented with the above screen.Template transitions require that you specify the correct side of the road in the lower left, then click the Grade or lane to alter, which is the first lane on the right, which is set to 13.73 according to the plans. To make sure the lane is fully expanded from the standard 12 to the 13.73 at station 400, it is necessary to set the "Begin Transition Station" to something less than 400, as shown. Then if this "expanded" lane width does not transition back to standard 12 width, but changes again, you must click on "Link to next transition" and leave the "End Full Template" and "End Transition" stations blank. Then you click "Add" again for the final segment, which would be entered as shown:



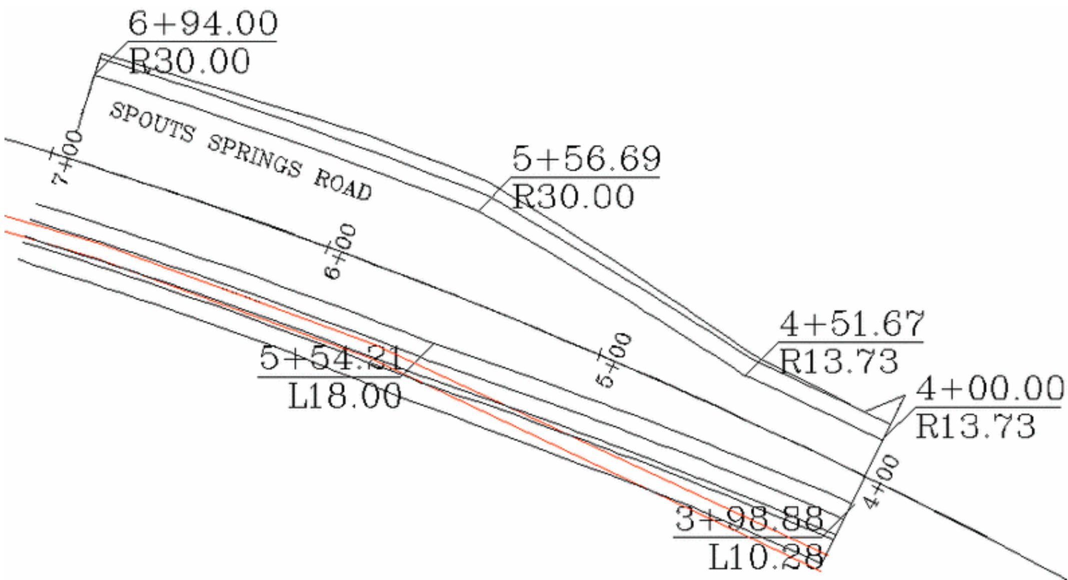
First, you specify "Side to Apply" as "Right", then click the pavement lane and edit it to 30', as shown above. Referencing the plan view drawing for Spouts Road shown above, you transition from station 451.67 to the new 30' road lane width at station 556.69 and hold that to the "End Full Template Station", which is 694.00. Then you can enter an "End Transition Station" just past the end of the key station range, which internally would transition the template back to a standard width of 12' at 694.01 (a moot point as the end of the project is station 694 for this exercise). The key to template transition is that it is designed to transition from normal to expanded or reduced dimension, then transition back to normal. It is ideal for use in passing lanes that appear and then transition back, but requires use of "Link to next transition" to handle a sequence of lane width changes as above. Therefore, where lane widths change often, and don't transition back to the normal template lane width, it is often best to use Template Point Centerline as the method of lane transitioning. We will apply that below to the ditch line.

When the template transition process is repeated for the left driving lane, you obtain a final Template Transition dialog as shown here:

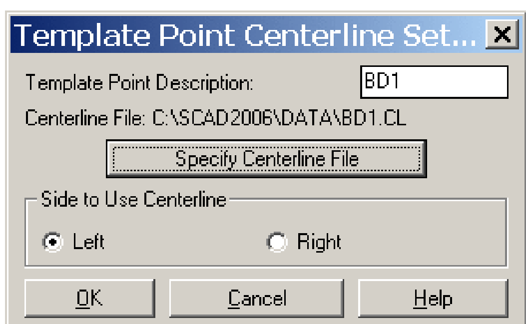


For the left side, the first screen just starts things up by establishing 10.28 as starting left side dimension, then the "Link to next transition" option is used, and the width of 18 is entered, transitioning to 18 at station 554.21 and

holding that to an end station of 764.34, transitioning "back" to 12 at the fictitious 764.35, well beyond the 400 to 694 station range of interest. When this template transition file is run in Process Road Design and Triangulate & Contour is turned on within Process Road Design, the output clearly shows that the lane transitions have followed the lane expansions correctly:

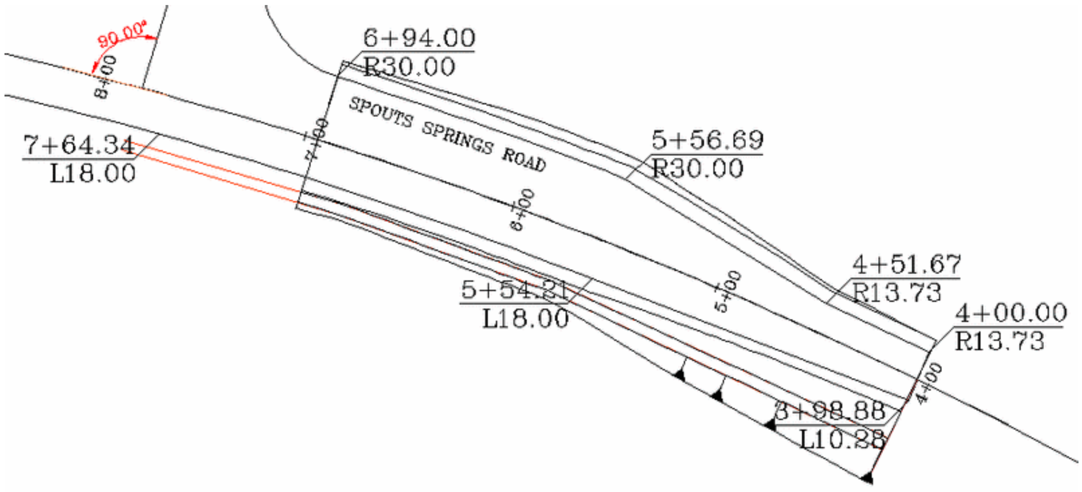


However, it is easy to see that the "design ditch" on the left side of the road, at 2' wide, did not conform to the special ditch which hugs the shoulder at station 7+00 but transitions to further off of the shoulder at 4+00. This special ditch is best handled with Template Point Centerline. To complete the special ditch design, use Polyline to Centerline File on both ditch polylines, calling the inside polyline BD1.CL and the outside polyline BD2.CL, as a reference to the ditch IDs, BD1 and BD2. You can give them a starting station of 0. The stationing of the ditch polyline does not matter, since only the coordinates of the centerline in the command Assign Template Point Centerline are used to determine the template ID position. Within Assign Template Point Centerline, Add each of the ditch sides as shown:

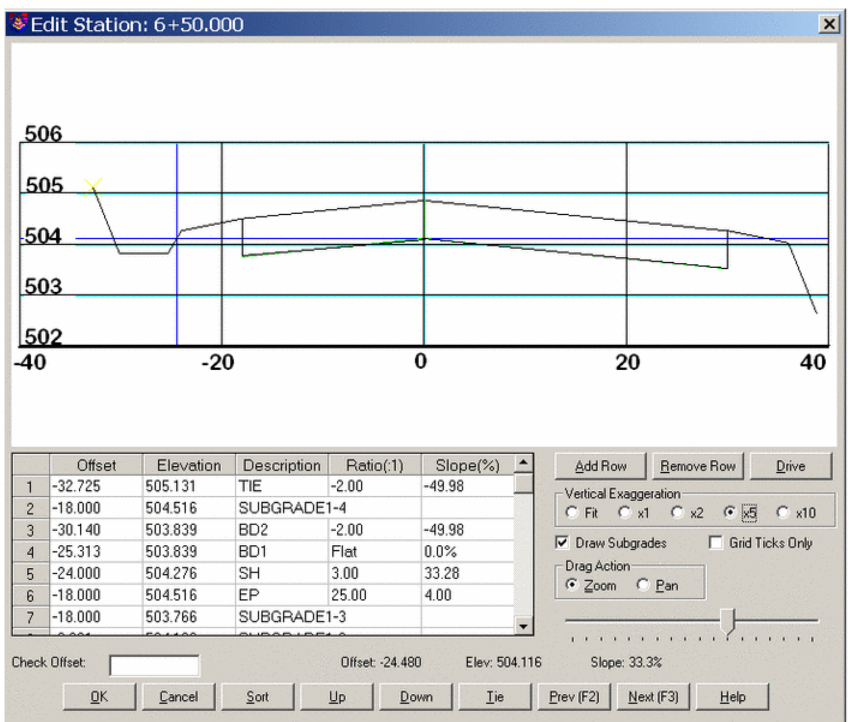


Note that if the ditch always exists on the left side, the ditch grades can be defined using the Grade button in Design Template, rather than using the Ditch feature within the Cut button. For final results, run the Process Road Design command using a combination of the Template Transition File and the Template Point Profile.

The end result is a final drawing that uses the Template Transition file to create the correct edge of pavement and uses the Template Point Centerline file to track along the correct ditch polylines. This is shown below in the final drawing of the 3D polylines generated by Process Road Design:

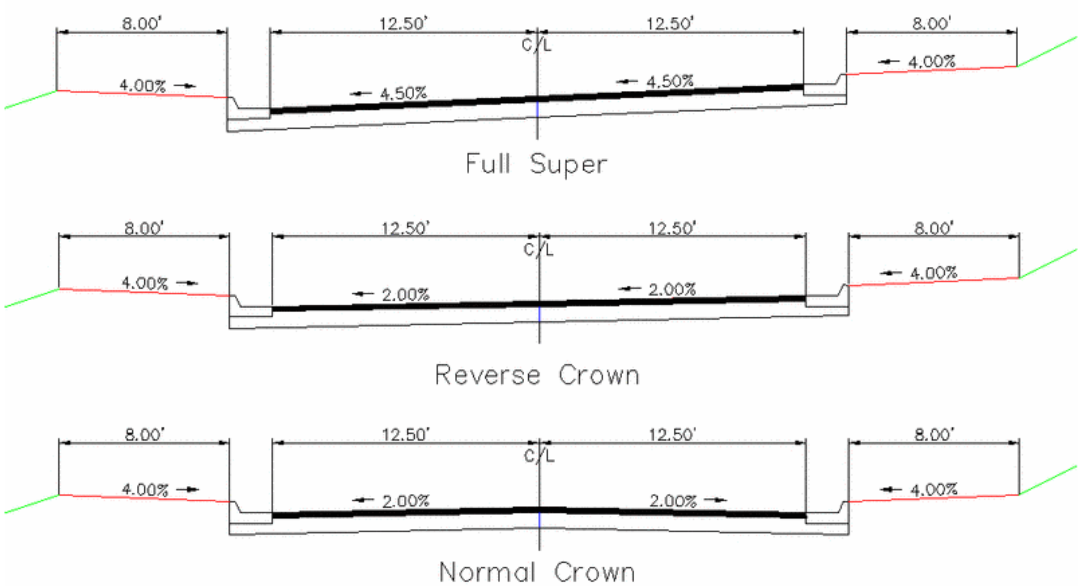


The actual slope to the ditch on the left is held at the design of 3:1, or whatever exists within the template from shoulder (SH) to base of ditch (BD1) in cut. Shown below in the Input-Edit Section File screen editor is station 6+50, where the ditch is designed very close to the shoulder:

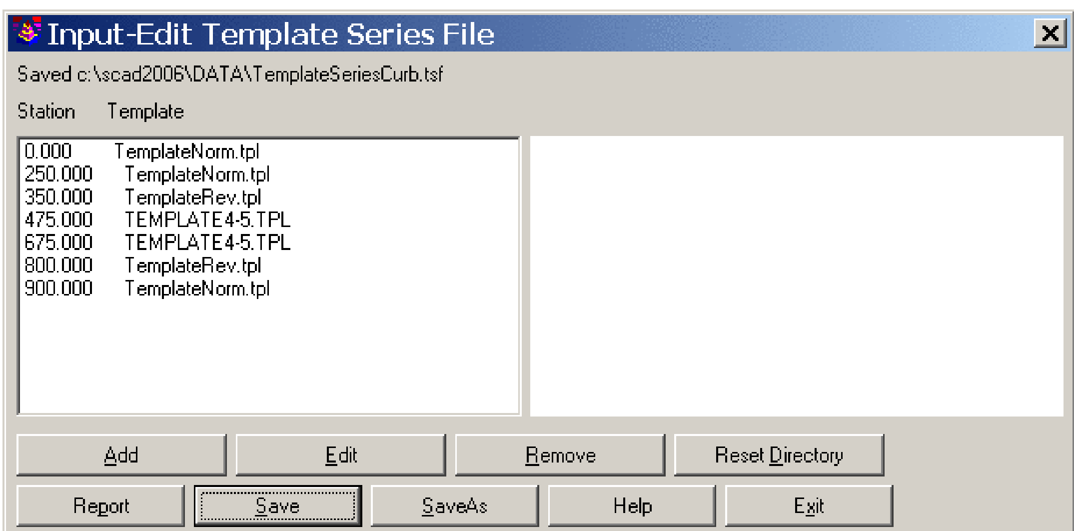


Note that the distance from BD1 to BD2 is irregular, based entirely on the plan view offset of the ditch polylines. Note also that BD1 to SH is 3:1, holding the defined slope. (The cursor position also can be used to verify slope of any portion of the section in "real-time".) Finally, note that the subgrade follows the widening and irregular position of the pavement lane EP for both left and right sides, since the subgrade offset from centerline was defined as EP.

Although superelevation can be handled by use of superelevation files, for most simple applications (2-lane roads in particular), a single curve with superelevation can be handled by a template series file, using only 3 templates: normal crown, reverse crown, full super. This is illustrated below, for a typical 2-lane road template:



The actual Template Series File will consists of 6 entries for one curve: Normal, Reverse, Begin Full Super, End Full Super, Reverse, Normal. You would only need to make one extra template, for simple roads, for every additional curve, for the full super condition, since normal and reverse crown remain the same. Note that the curbs, even on the high side, can be designed to slope downward and catch the shoulder drainage in Design Template by use of "special slope" of -1% in the curb design, or by entering a value for the added "Drop" across the gutter portion. Both methods create a downhill slope to the face of curb. So the above project might be designed as shown below in the Input-Edit Template Series command:



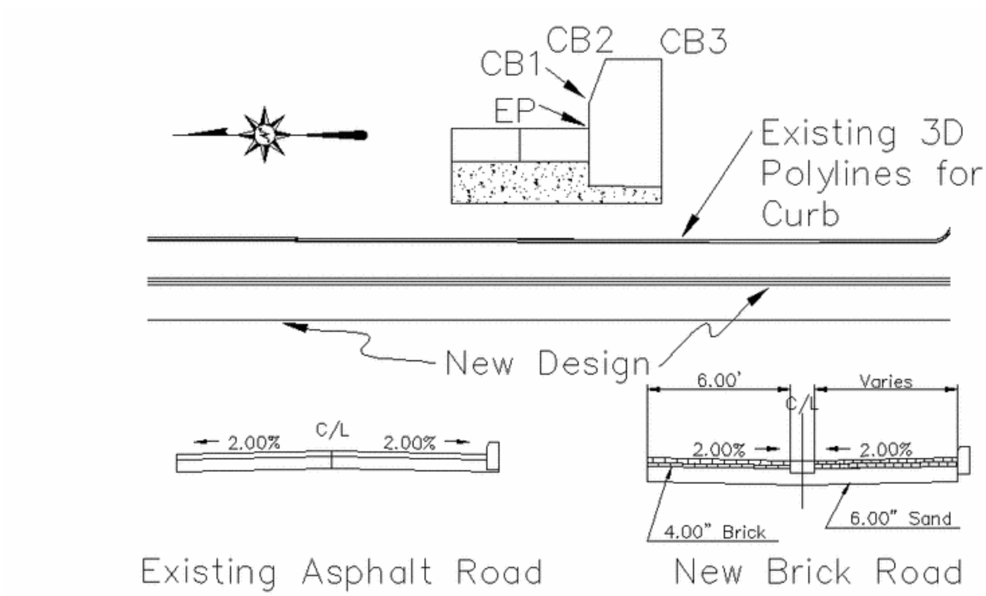
Note that beginning and ending stations are not necessary. If station 0.00 was omitted, Process Road Design would use the normal template in any case from station 0 to 250. Similarly, Process Road Design will use the normal template going forward from station 900 automatically.

Review of 2 Methods of Matching Portions of Existing Roads

There are two main techniques for tying new template designs into existing roads, which may apply to road expansions, urban re-paving, grade improvements and other renovation projects. As more and more roadwork involves road improvement rather than new road development, these techniques become more useful and critical to master.

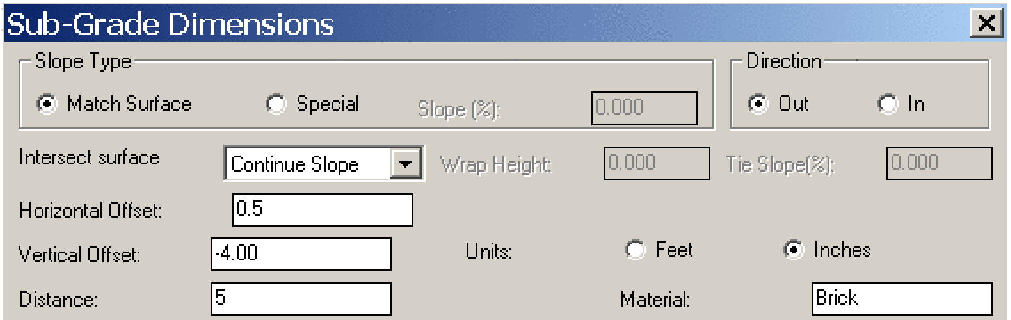
The two techniques are: (1) Use of Template Point Profile and Template Point Centerline files to match existing conditions on portions of roads that do not change, and (2) Use of the "As-Built" cross section feature as one of the input files. An advantage of the As-Built method is that you can insert section points with special IDs for special features, whereas the Template Point Profile and Template Point Centerline methods must follow template IDs that are found in the original, main template design file. But the Template Point or "string" method allows for calculating sections at any interval, while the As-Built section method will revise final sections only at stations found in the As-Built section file.

Consider this alley-way, which consists of a Belgian block style curb (no gutter) that is already in place. The plans are to remove a crowned asphalt alleyway and put in a bricked alleyway on sand, with a central, "depressed" rock drain of 1' width, to avoid water draining against buildings that abut the alley. But the design must match an existing "Belgian block" style curb on the right side of the road, which will not be removed.

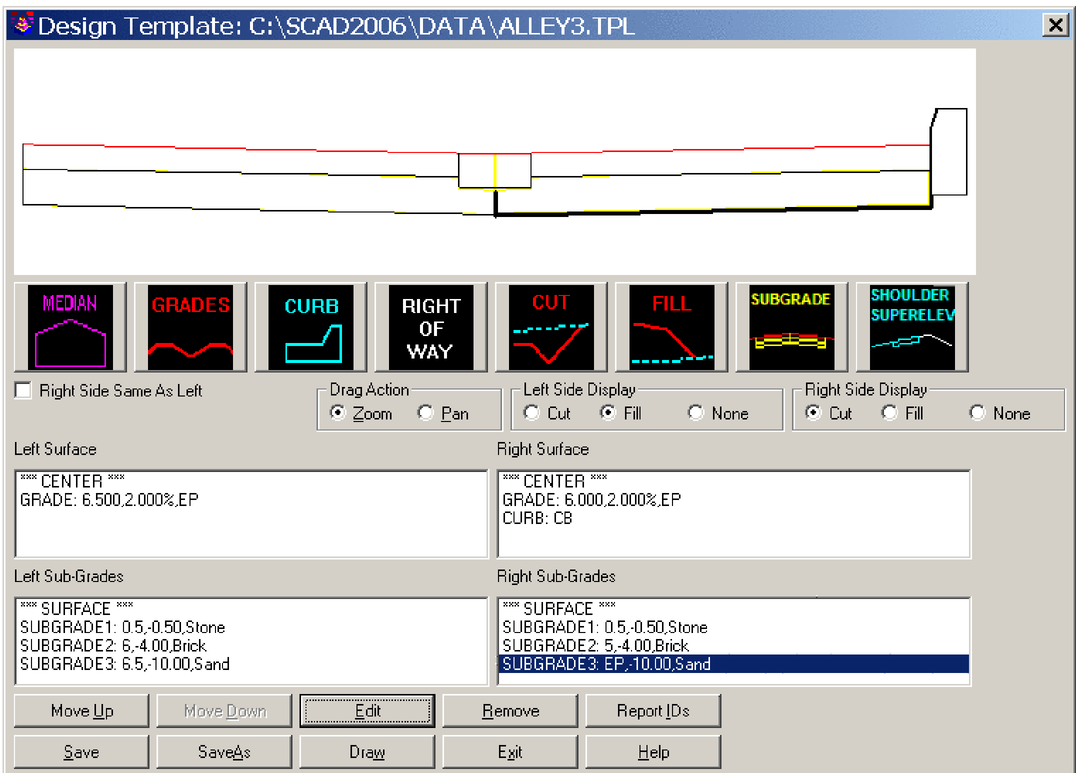


There is a new profile design involved, and a new template. However, the right side of the template will meet the exact grade and offset of the in-place curb, which has been surveyed as back of curb (CB3). Then the command Offset 3D Polyline was used to create the face of curb at EP=CB1, and to create the inside top of curb (CB2). Because of the symmetry and consistency of the curb, only the back of curb needed to be surveyed to hold the existing curb feature in place within Process Road Design. From that survey, the 3D Polyline for the EP is derived, which will be used for Template Point Centerline and Template Point Profile.

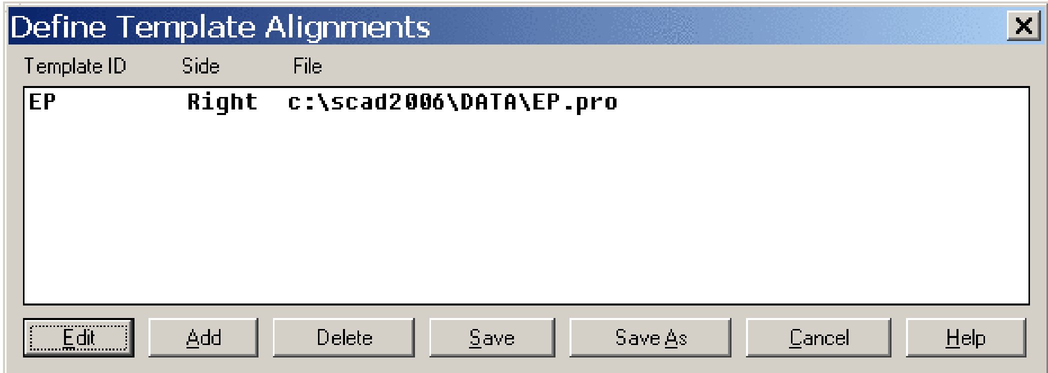
Features such as curbs and medians can be designed once within Design Template and then saved as curb or median files, then re-loaded and used in other templates, and applied to the left or right side of the template as desired. The central rock median of 1' total width can be constructed as two subgrades, one on the left side of 0.5' width and one on the right side of 0.5' width. The brick portion can be designed as a 4" thick subgrade as shown below. On the left side, you would need to use the "Straight Up" method of closing the subgrade surface. On the right side, you can use "Continue Slope". When using Continue Slope, it is best to underestimate the length needed to contact the next surface (the right curb), so continue can do an "extend" and find it. If you make the length too long (e.g. 6', which catches the curb which itself tilts back -2%), the program will not trim and will draw the subgrade to the back of the curb. Note that the vertical subgrade depth can be entered as 4 or -4. Both are accepted.



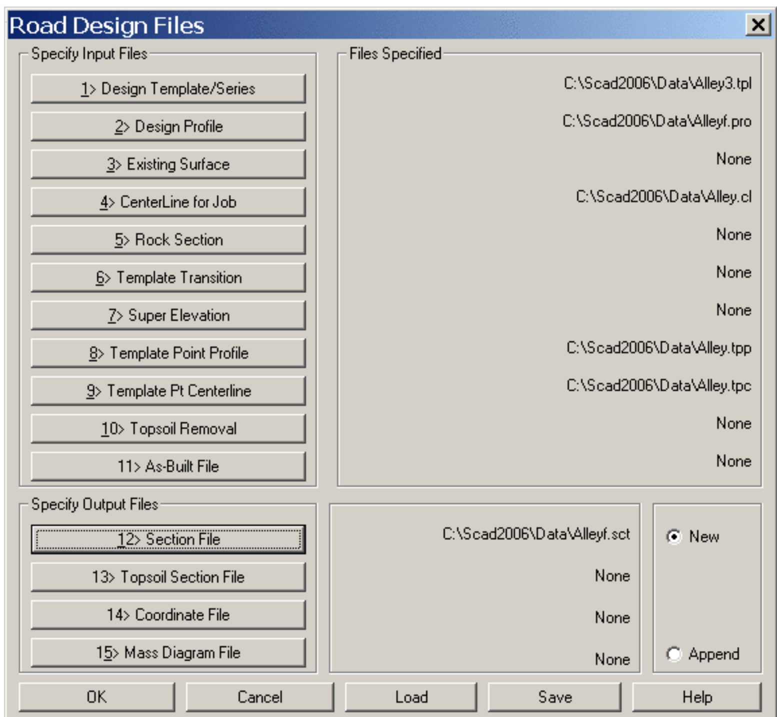
Be sure to define the sand subgrade on the right side (lowest subgrade) to have a distance of EP, a flexible distance that follows the precise offset of the EP "ID", which will be assigned to follow the face of curb template point profile defined by CB1 above.



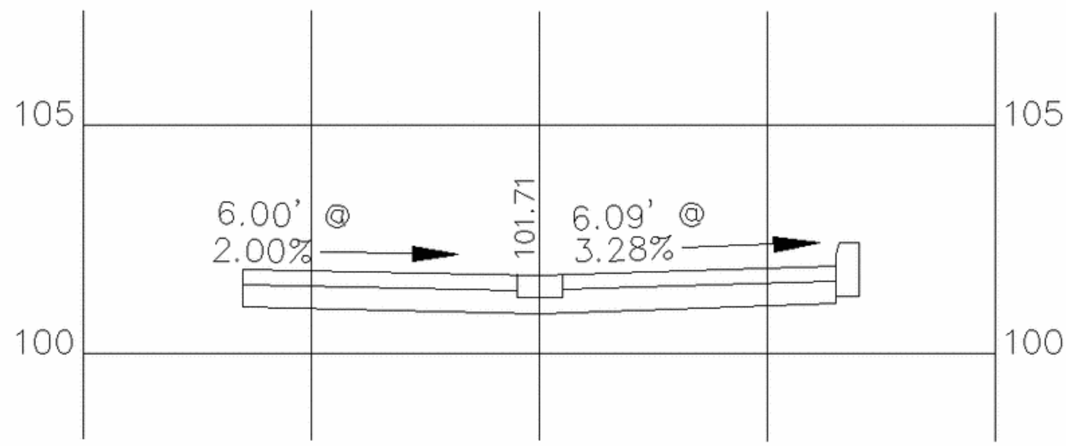
The next step is to set up the face of curb 3D polyline as a template point centerline and template point profile assigned to "EP". First you must do Polyline to Centerline File, pick the inner 3D polyline which is face of curb at proposed road level. Then you must do Profile from 3D Polyline and make a profile for the "EP". Then you assign this centerline and profile to the appropriate ID (EP) to force the curb to contact the correct curb position and elevation. The curb defined in the template matches the pattern of the in-place curb, so by setting EP to the correct template centerline and profile, the curb will "follow" at the correct position. The stationing used for the template point centerline is not critical to the calculation. However, the profile stationing much match and reference the centerline stationing. Therefore, when doing the command Profile from 3D Polyline, answer Yes to the question: "Station by another reference centerline [Yes/<No>]:". Making the Template Point Profile is always best accomplished by this method of Profile from 3D Polyline, referencing the design centerline. The Template Point Profile (and Template Point Centerline) would appear as shown here:



The files in Process Road Design would be set up as follows:

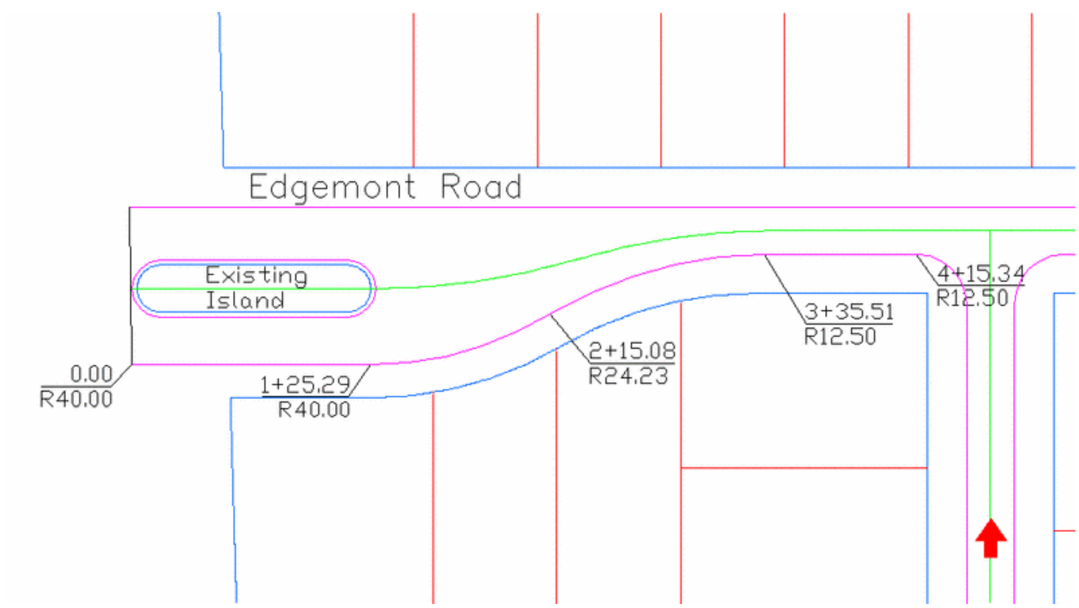


Note that no existing surface file is needed to compute final cross sections from as-built (straight wall on left of alley) to as-built (existing curb on right of alley). A final section is plotted below, showing the unique slope and lane distance determined by the as-built centerline and profile files that control the edge of pavement, and by extension, the curb, which continues with fixed dimensions from the edge of pavement.

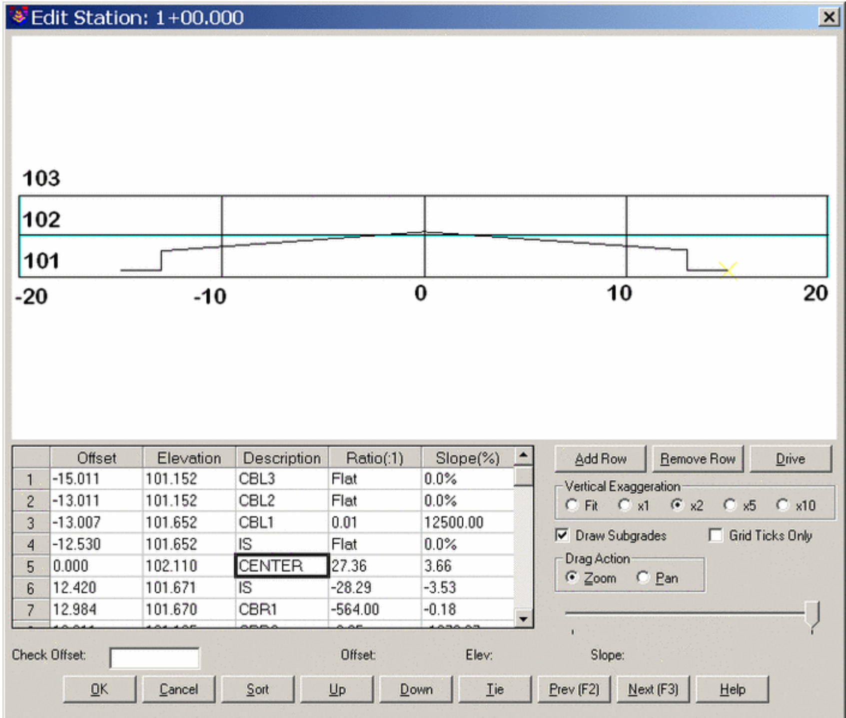


Station 0+50

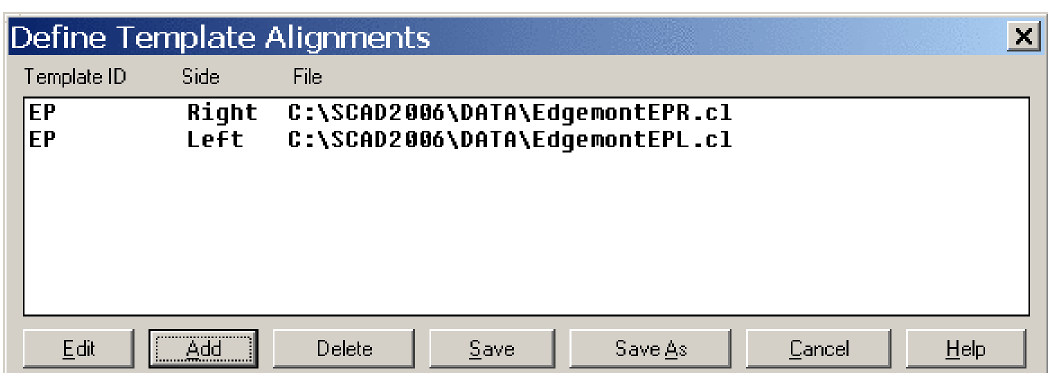
A second method of doing as-built road design is to use the as-built cross section method. Whenever as-built cross sections are specified as part of the input files in Process Road Design, and then referenced for use on the Additional Road Design Parameters screen within Process Road, those offset IDs that are referenced will be held. Any matching IDs or new IDs found in the as-built cross sections will be substituted for the designed IDs within the final sections. In the example below, it might be proposed to redesign Edgemont Road from a roadside ditch road to one with a curb and gutter as well as sidewalks. However, the designer might want to keep the existing central median, already curb and gutter with plantings.



This example raises the challenging issue of inserting special interior points with new IDs into a set of design cross sections, through a length of about 125 feet of road. If a cross section of the island is taken through station 1+00, it might have the following ID points:

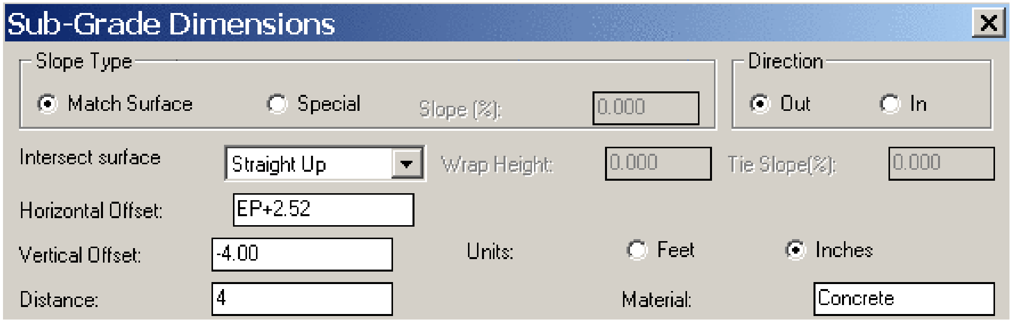


This cross section could then be part of an as-built cross section file (.SCT) which can be recorded at any desired station interval, the smaller the interval, the greater the accuracy. Now if the actual road template is defined as EP for edge of pavement and standard CB for curb, with CENTER for the centerline position, Process Road Design will substitute the As-Built File CENTER ID for the one calculated by the program, and will add in all the unique IDs from the cross section file, from -15.011 left to 15 right. Interestingly enough, this Edgemont Road example would also require a Template Point Centerline for the left and right edge of pavement, to pull the paving edge out to the expanded road dimension, which doesn't taper to normal until station 3+35.51. It would not require a Template Point Profile, so long as the road maintained a consistent design slope from centerline. When using Template Point Centerline, you need to turn the edge of pavement polylines into centerline files. Before doing so, test each polyline with the command Reverse Polyline (within Polyline Utilities under Edit) to verify that the polyline is drawn in the correct direction, as shown by the phantom arrows. The file Template Point Centerline elements might appear as shown:

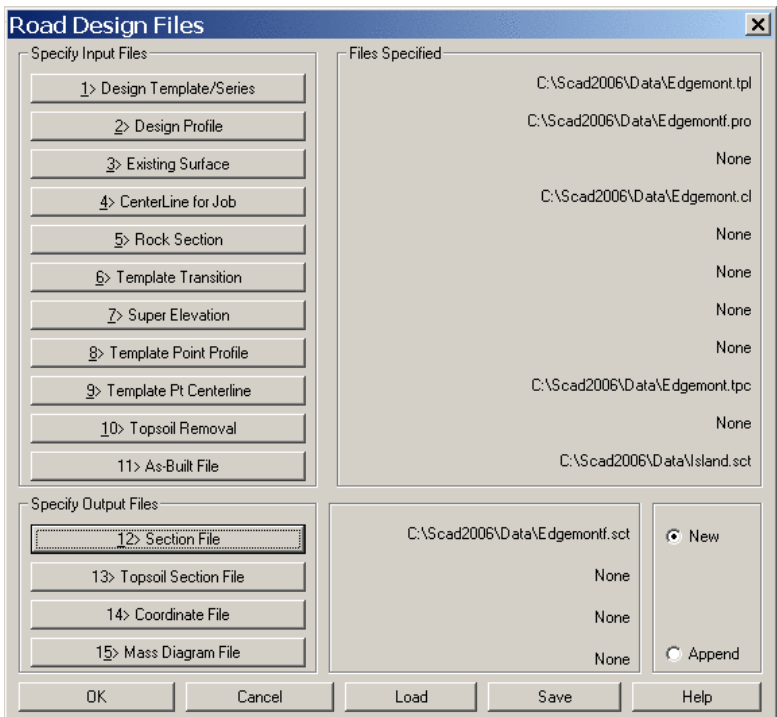


Be aware that a subgrade such as a concrete sidewalk, if it is to be placed behind the curb, must reference the curb or the edge of pavement ID for positioning, whenever the edge of pavement offset is changing based on use of a Template Point Centerline or As-Built cross section file containing duplicated IDs for edge of pavement. You can specify an offset for the sidewalk in the Subgrade option within Design Template, as shown below. The "2.52" offset

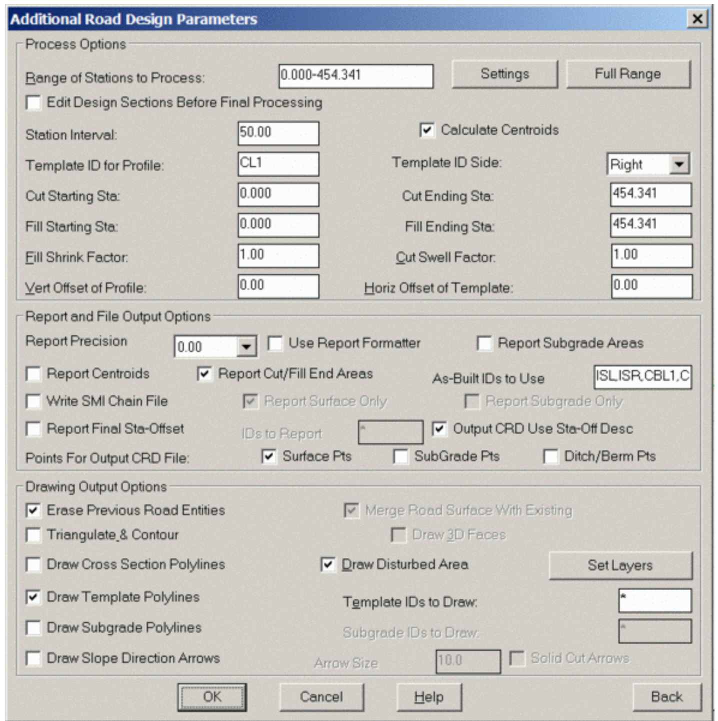
was used to move past the tilting edge of the back-of-curb, which slightly exceeds 2.50.



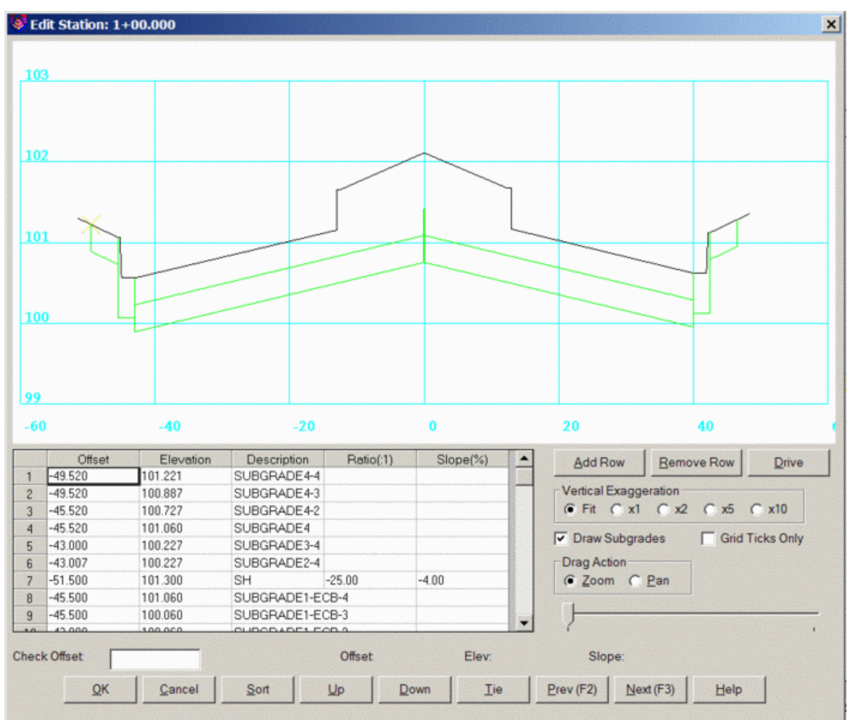
If the Island.sct file is the as-built cross sections, the entire input screen for the Edgemont Road project might appear as follows:



In the next dialog, fill in the descriptions for the section points in the As-Built IDs To Use field.



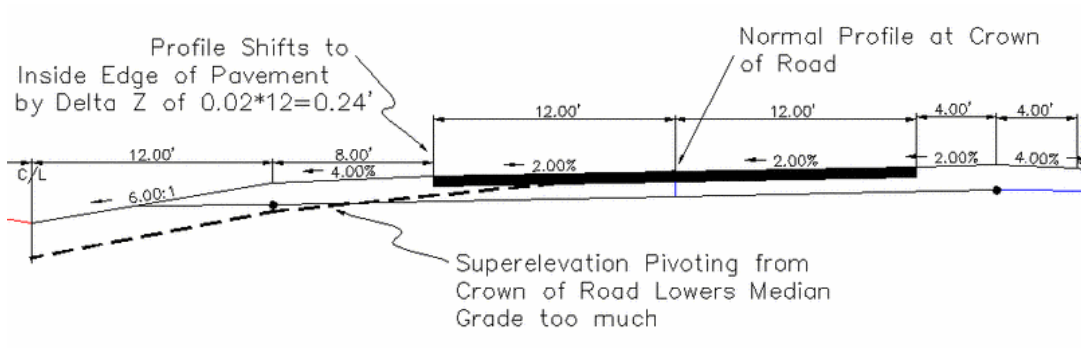
Here is the resulting output section file showing the combination of the design template with the as-built section points.



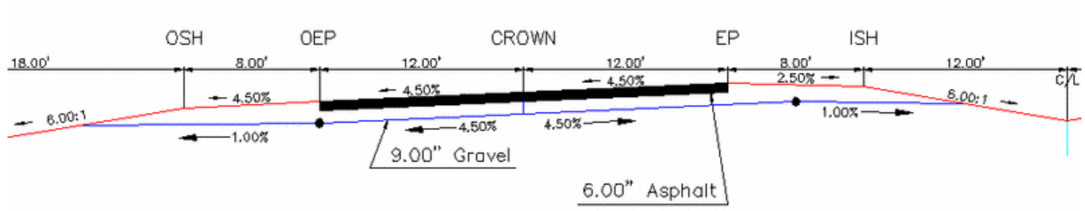
Example Divided Highway with Special Super Elevation Treatment

Divided highways such as 4-lane highways with a central depressed, grassy median are among the most challenging roads to define as templates, especially when accurate subgrade elevations and quantities are involved. Rules for

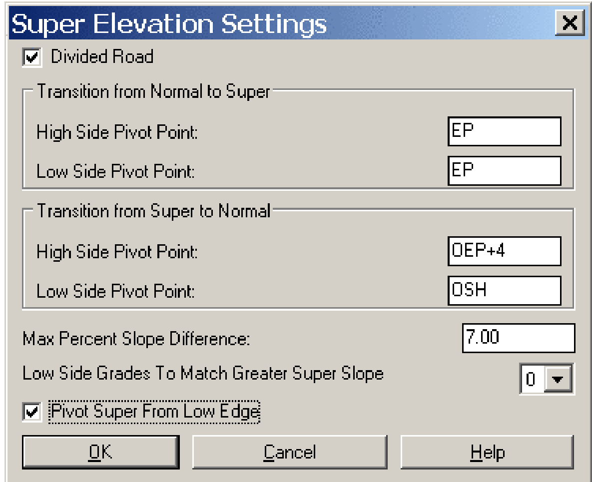
superelevation and subgrade pivot points must be applied. And most divided highways do not use the centerline as the profile and require shifting the profile elevation to a specific template ID, like the inside edge of pavement or crown point for each side of the highway. This shifting occurs within Process Road Design. Furthermore, many highway departments have complicated rules for the profile grade. One such rule is that in superelevation, when the pivot lane reaches reverse crown, the profile moves from the crown of the road to the inside edge of pavement. Whatever the delta Z between the crown profile grade and inside edge of pavement profile grade is at reverse crown, this delta Z is subtracted from the profile grade and determines the profile of the inside edge of pavement from reverse crown through full super and back to reverse crown again. This typically improves drainage within the median portion, since a steep superelevation pivoting from the crown of the road can either reduce the median depth, or force the median too low. This is illustrated in the graphic below. Such challenging highways can be designed using special features within Design Template and Process Road Design.



The divided highway template itself can be quite complex. Let's review the requirements of our template below, first left side, then right side, in superelevation of 4.5%.

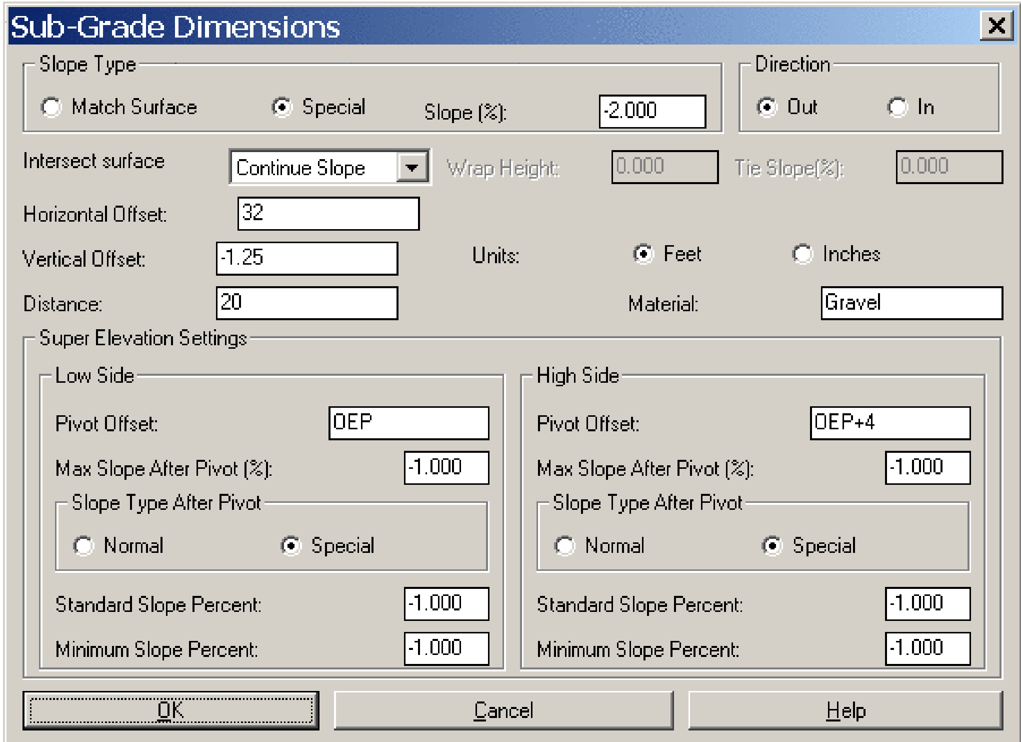


The main criteria for the design is that the pavement lanes are 12' wide, with 2% slope from the crown point in the middle (except in superelevation). On the interior high side of superelevation shown above, the grade breaks off at the EP or inside edge of pavement, and the maximum algebraic difference is 7%. So at 4.5% superelevation, the normal 4% downhill shoulder slopes instead at 7%-4.5%=2.5%, as shown. This part of the template behavior is controlled by the Superelevation Shoulder button within Design Template, with entries as shown here:

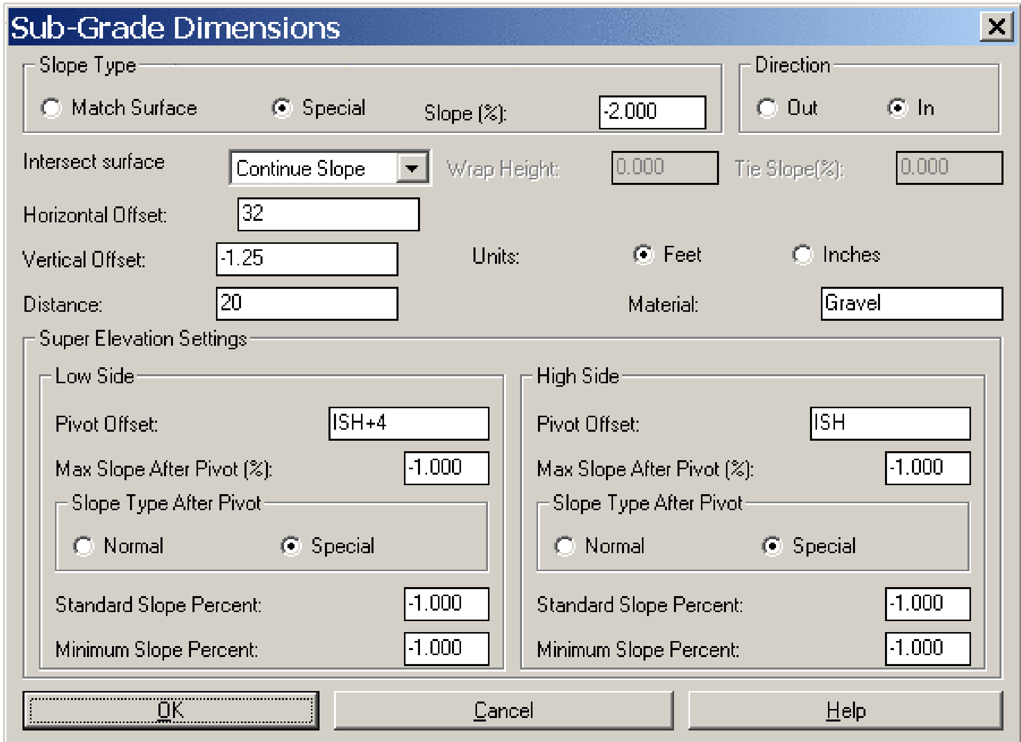


Note that the Super Elevation Settings dialog treats the "interior" of the road in the upper part, and the exterior of the entire road (like a 2-lane road) in the lower part. So the "Low Side Pivot Point" under the lower "Transition from Super to Normal" is where, walking from the middle of the road towards the left, super ends and normal slopes resume. That is set to OSH, or the outside shoulder position, the goal being to slope the full shoulder with the superelevation on the lower outside shoulder lane, then resume normal (non-super) slope at the 6:1 "recovery zone" slope. The entry of OSH as Low Side Pivot Point for Super to Normal controls that. In the upper part of the dialog, the inside "Transition from Normal to Super" sets the Low Side Pivot Point at EP. So at EP, walking from the template center left towards the left side of the road, normal ends at EP and superelevation begins. So the median upslope of 6:1 is normal, as is the shoulder, the super starts at EP. But because the 7% maximum percent slope difference is active, the shoulder can't remain at 4% but goes to 2.5% leading to the 4.5% superelevation. When super subsides to 3% or less, the shoulder would be normal at 4% as specified in the template design in this case.

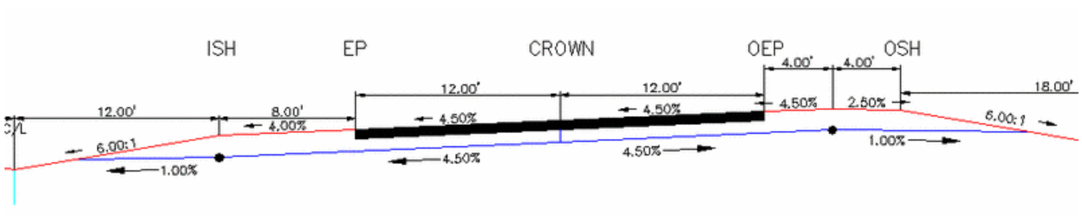
Referring to the graphic above showing the left side of the divided highway, the gravel for the shoulder is shown running out to "daylight" on the outside recovery zone and on the inside median slope. However, to reduce quantities of stone, the stone runs at a uniform slope of -2% in normal crown, or matches superelevation, but pivots to 1% downhill at the outside OEP and 4' past the inside EP. This is accomplished through the subgrade entry dialog. First, the outside subgrade:



Note that the normal slope of the stone subgrade does not follow the surface but stays at the "special" slope of -2%, matching the surface always only beneath the asphalt portion within the pavement zone. For divided highways, it is always necessary to do at least 2 subgrades for each material: one from the crown or middle of the road "out" to the outslope (as above), and one from the crown or middle of the paved portion in to the interior. Since the crown of the road on each side of the highway is 32 feet left of the center depressed median position, the horizontal offset for the "out" position is 32. Enter the vertical offset as the entire distance from the horizontal offset down to subgrade bottom. In this way, any other thinner subgrades above are deducted from total subgrade quantities of the grade under consideration. If the goal is to "force" a -1% slope in both normal crown and superelevation, then set the Max Slope After Pivot(%) to -1%, and click "Special". Then set both Standard Slope and Minimum Slope Percent to -1%. This ensure that -1% will be used at the pivot offset of OEP, or as specified. Apply this to both subgrades ("in" and "out" from horizontal offset 32). If you simply entered -1% for the Max Slope After Pivot(%) and clicked Normal, slopes on the low side would break over to -1% but slopes on the higher side of each superelevation lane (beneath inside shoulder on the left, outside shoulder on the right) would continue on at the super slope and not break off. You must use the "Special" setting. The low side shoulder for the inside portion of the left side of the road is specified by the "In" subgrade, in this dialog:



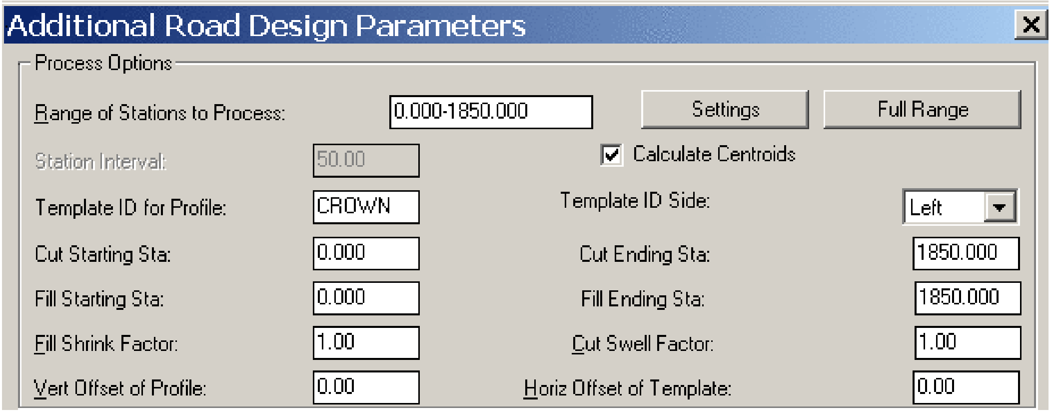
The pivot point for the subgrade on the inside left of the template is ISH+4, or 4 feet from inside shoulder to inside edge of pavement, the +4 being the direction walking out from the middle of the template in all cases. The right side of the template is shown next:



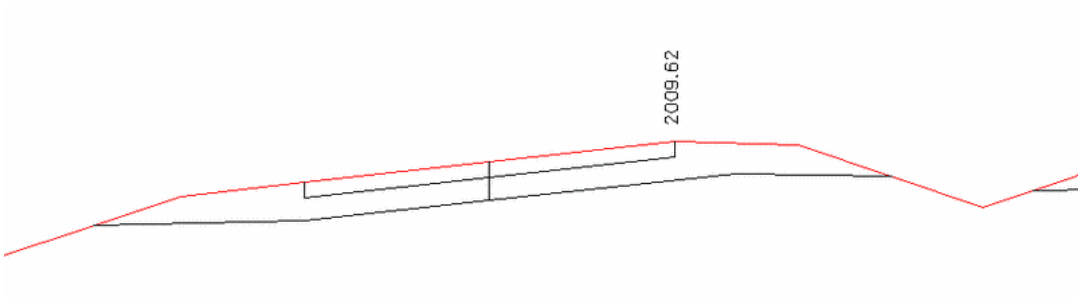
On the right side, the high-side subgrade pivot in the "out" direction, walking from the middle of the road outward, is OEP+4. On the right side, the high-side subgrade pivot in the "in" direction is simply ISH, as shown. So the controls exist to specify critical break points on subgrade and surface grades using Design Template. Whether this is the best design can be debated, but the controls are there to create surface and subgrade slope breaks and grade changes.

Referring to the Super Elevation Settings dialog above, the key to setting the superelevation of the divided highway to the inside edge of pavement at reverse crown (minus the 0.24 delta Z from profile grade to inside edge of pavement grade) is to click on the option, "Pivot Super From Low Edge".

Now you must run Process Road Design, using this template, to produce verifiable final cross sections. Set the Process Road "Additional Parameters" dialog such that "Crown" (or whatever ID is used for the center crown point on each side of the road) controls the profile grade.



The final sections that are produced will shift the profile grade to the inside edge of pavement from reverse crown to reverse crown through superelevation, adjusted -0.24'. A final section is shown plotted below as drawn using Draw Section File:

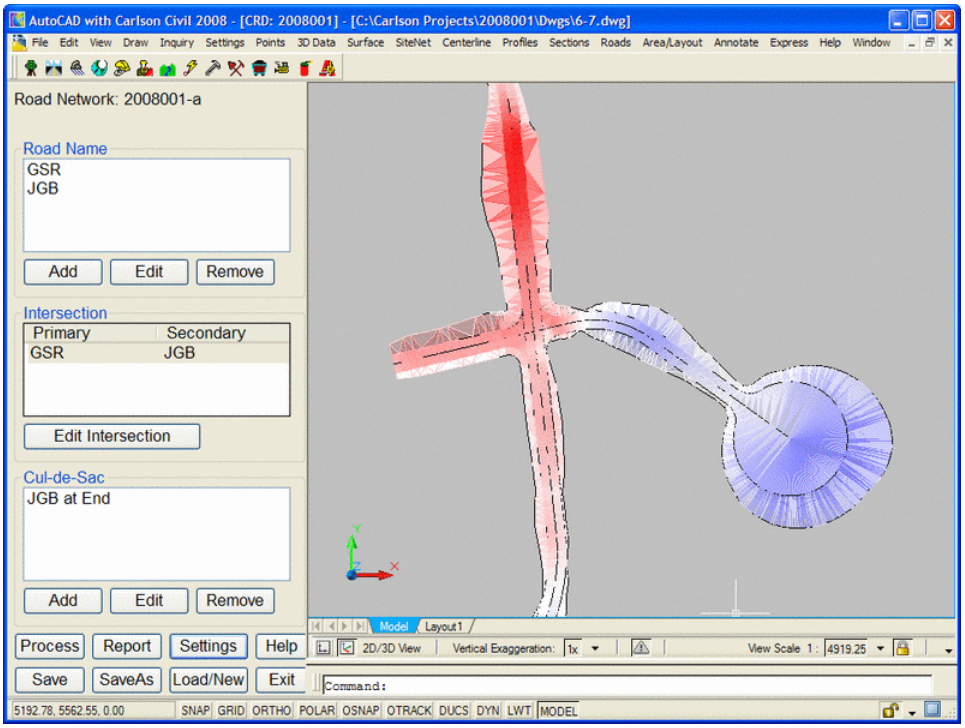


Pulldown Menu Location: Roads
Keyboard Commands: eworks
Prerequisite: Profile file and template file
File Names: \lsp\eworkd.lsp, \lsp\eworks.arx, scadewrk.dcl

Road Network

Function

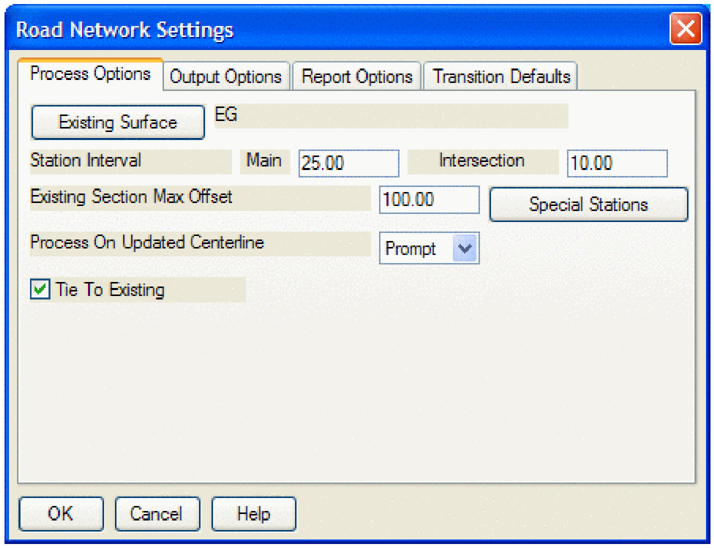
This command synthesizes road network design for subdivisions and commercial and industrial sites by enabling interactive 3D design of all road centerlines, profiles and templates, including cul-de-sacs. A docked dialog on the left of the screen identifying the existing DTM surface and all road files combines with an active CAD screen and command line. You can save drawings and run virtually any standard Autocad command while within the docked dialog. Once the user identifies all centerlines involved, the program detects intersections and end segments suitable for cul-de-sacs, and through user input of design parameters for cul-de-sac dimensions and intersection transitions, the program will process the complete 3D design, with output options including cross sections, 3D faces, TIN files and contours. The many roading files involved in a road network design are all saved to an "RDN" file that can be recalled, modified and re-processed.



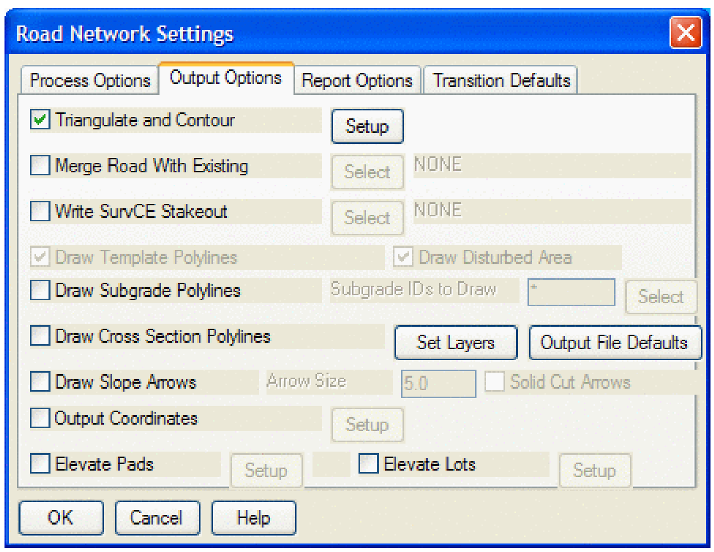
Procedure

Step 1: Start the Road network command, and either create a new Road Network (.RDN) file, or select an existing one. If you have previously run Road Network with the current drawing the Road Network docked dialog will open with the last Road Network (.RDN) file you worked with. If this happens, but you prefer to create a new Road Network (.RDN) file, or use another existing Road Network (.RDN) file, click the Load/New button at the bottom of the Road Network docked dialog. If you wish to make a copy of the current Road Network (.RDN) file, click the SaveAs button.

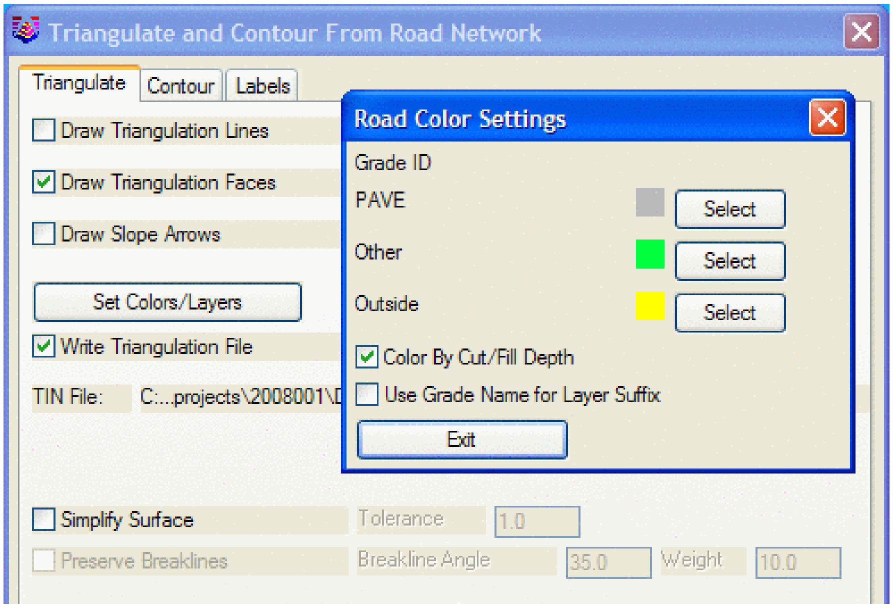
If this is a new Road Network (.RDN) file, the first thing to do is to review the Settings, and be sure to have the Existing Surface set. Click the Settings button at the bottom of the docked dialog. The Road Network Settings are divided into four tabs, Process Options, Output Options, Report Options, and Transition Defaults. On the Process Options tab, pick the Existing Surface button to set the target surface for the road network to project side slopes to. Either a TIN or an FLT file are accepted as valid surfaces, both of which can be made within the command Triangulate and Contour. For speed, it is recommended that the binary TIN file format be selected. This tab is also where you set cross section processing options, including station interval through the main part of the centerlines, as well as a different value to be used going through intersections, the maximum offset from CL for existing sections to be sampled, and any special stations that you want processed. Process On Updated Centerline sets how RoadNet responds to changes in the centerline. The Network can be reprocessed Automatically, not at all, or the user can be prompted that the Road Network needs to be reprocessed. Tie to Existing controls whether or not the side slopes specified in the template (.TPL) file are applied or not.

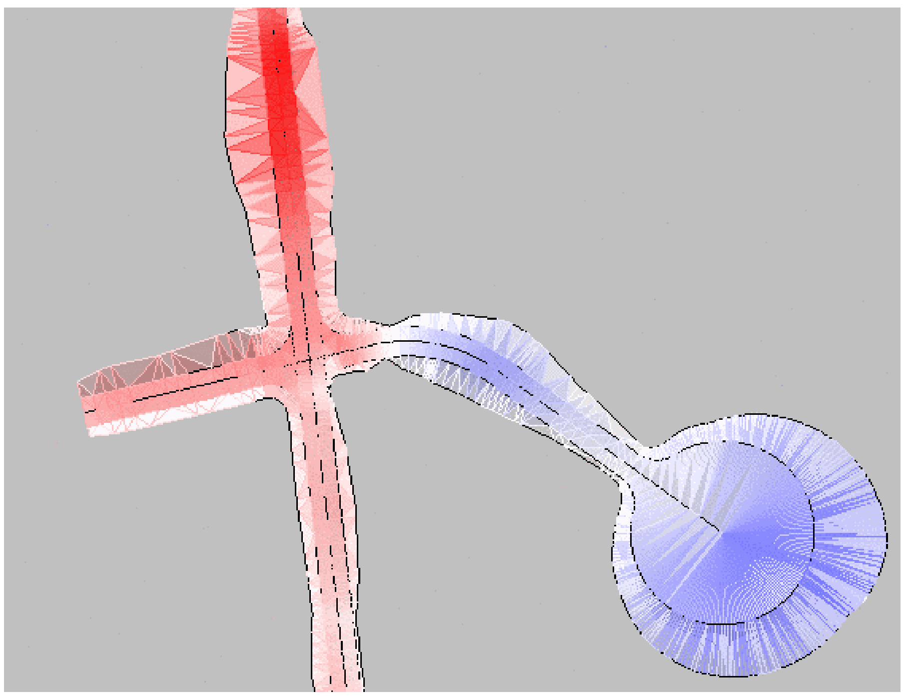


On the Output Options tab, if you want the Road Network to generate a Surface, check Triangulate and Contour and click the Setup button to specify the Surface name and other options.

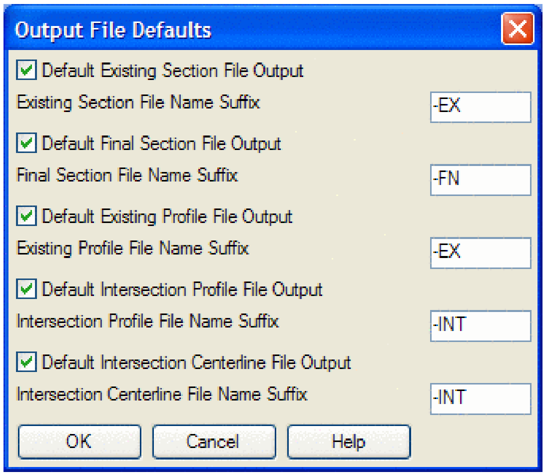


If the option to Draw Triangulation Faces is selected, the Set Colors/Layers button is selectable. The components of the templates used are shown here and their colors can be controlled. The Color By Cut/Fill Depth option colors the Triangulation Faces with a range of Reds and Blues to show areas and depths of Cut and Fill for the proposed Road Network.





The Output File Defaults button is where all Output File names are set.



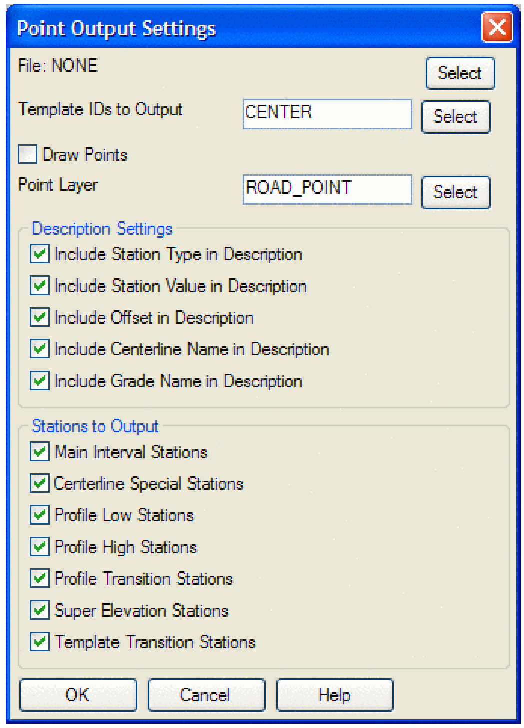
The **Merge Road with Existing** option gives the ability to merge the Road Network surface into the target existing surface to create a new surface that essentially depicts the entire site after the Road Network is constructed.

Write SurvCE Stakeout creates a SurvCE Stakeout (.RNF) file that contains all of the information for the entire network, including all centerlines and profiles. This file can be directly loaded into Carlson SurvCE data collection software for unlimited field stakeout of the road design.

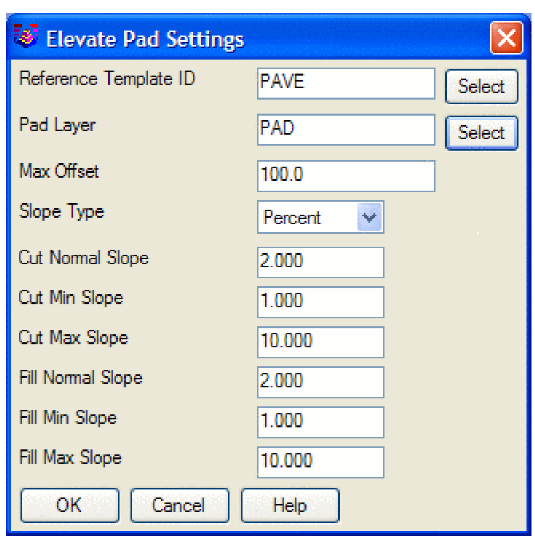
There are also options to **Draw Template Polylines**, **Subgrade Polylines** and **Cross Section Polylines**. These are typically used to generate additional surfaces for modeling, stakeout or machine control purposes.

Draw Slope Arrows creates arrows in the drawing to show the direction of each triangular "plate" in the Road Network TIN. This can be helpful to visualize where water will be flowing.

Output Coordinates generates Carlson Points for a wide array of critical locations in the Road Network. The dialog box to make selections of the desired locations for points is accessed through the Setup button.

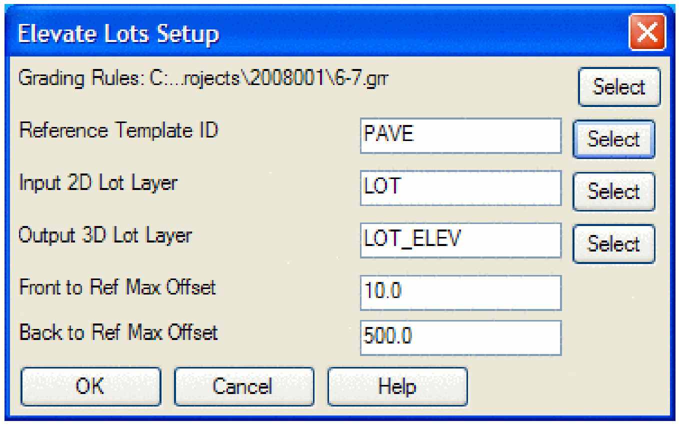


Elevate Pads adjusts the elevation of closed polylines within a specified proximity of the Road Network, relative to the elevation of a specified point on the Road Template (Reference Template ID). In the dialog below, all closed polylines on layer PAD that are within 100 feet of the road will have their elevations set based on a 2 percent grade up from the PAVE Template ID point, in either Cut or Fill conditions. In future earthwork balancing adjustments, the polyline can be adjusted a maximum of up to a 10 percent grade or down to a 1 percent grade from the Reference Template ID.

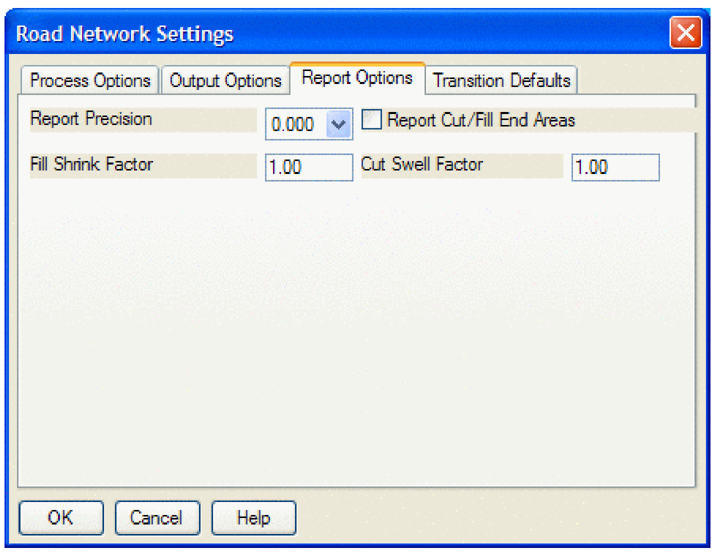


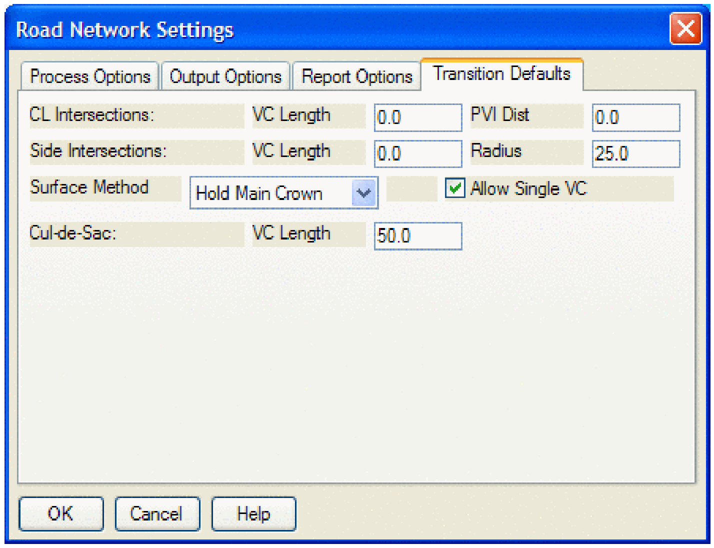
Elevate Lots is a similar function to Elevate Pads, but acts on a set of grading rules which are stored in a Grading Rules (.GRR) file. For detailed information on Grading Rules, please refer to this topic in the Help on the 3D Data

menu.

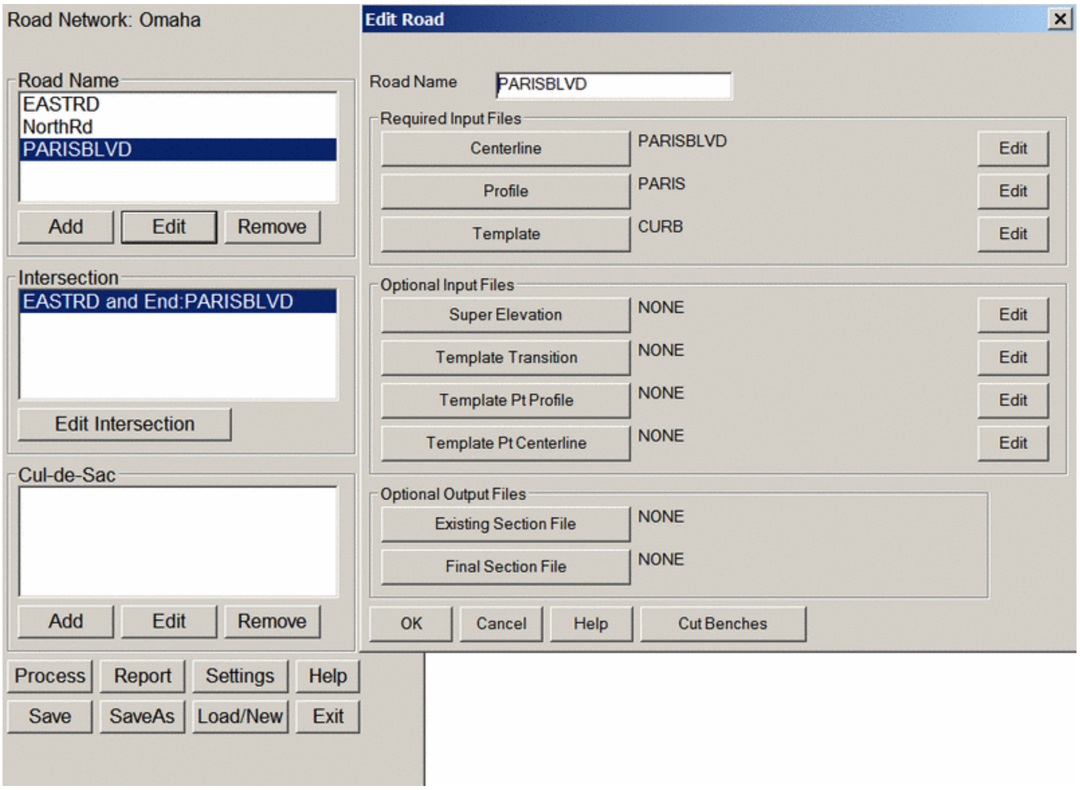


Review the Report Options and Transition Defaults tabs as well.

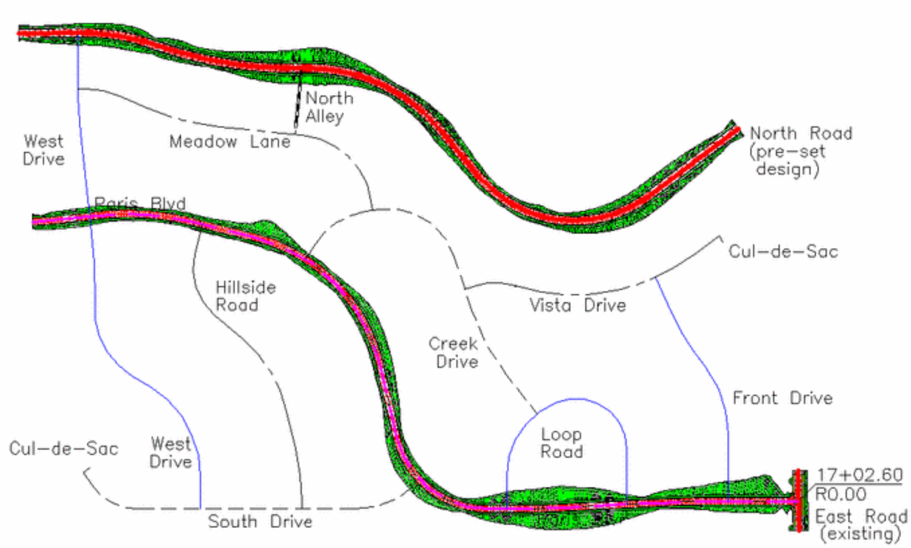




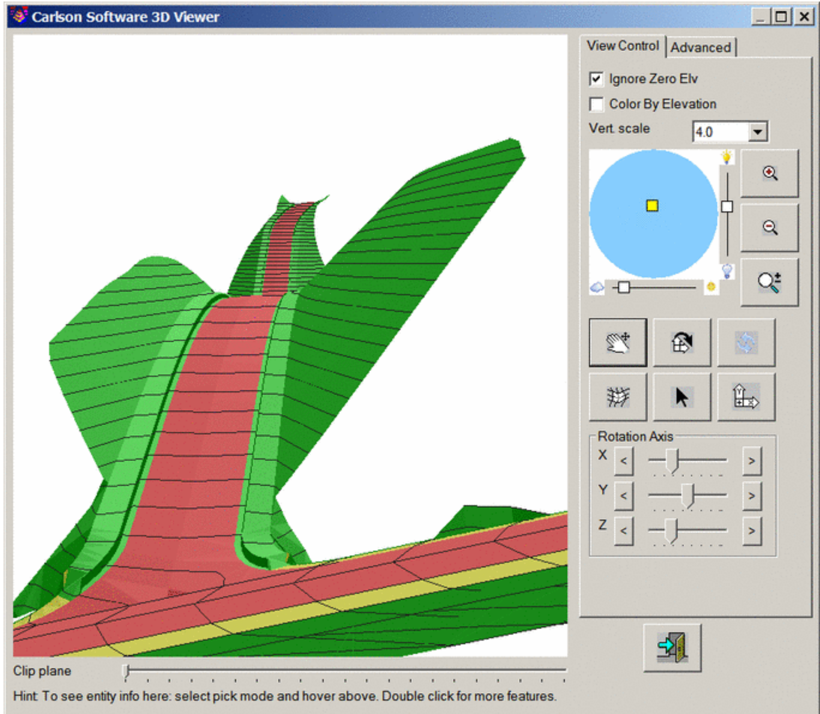
Step 2: Back in the main dialog, click "Add" in the upper left "Road Name" portion, and identify all of the main road and secondary (intersecting) road centerlines. Referring to our drawing above, we could do the full design at once or start by identifying North Road and East Road as the main roads and Paris Boulevard as the first secondary road. Note that centerlines may be picked as polylines or loaded as centerline files. All centerlines (horizontal alignments) must have, at minimum, an associated profile (vertical alignment) and an associated template. These must be pre-designed and stored, ready for recall, before executing the Road Network command. In the Road name dialog portion, select a road and click Edit to review the files. Note that by selecting Paris Boulevard and East Road, the program automatically detects the first intersect. As you follow the design below, you will see that we follow the hierarchy of the road precedence as outlined in the graphics. At every intersection, there needs to be a primary controlling road (template cross slopes are held) and secondary adjusting road (centerline profile adjusts to template of primary road at some transition distance).



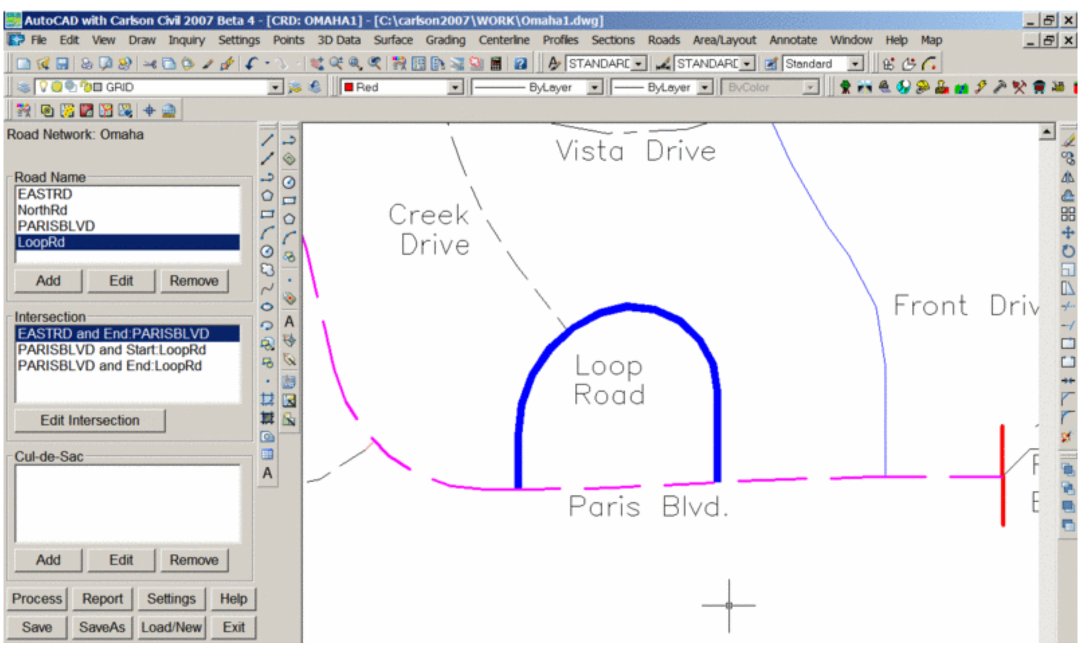
Step 3: Click Process to compute the design. Based on the settings in the above "Settings" dialog, the program will Triangulate and Contour and create the drawing shown below. If you edit any road feature or dialog entry and click Process again, the program automatically clears the last Triangulate and Contour drawing and creates a new final design drawing. In this way, you can trial-and-error your design for all roads, or build the design in stages.



Viewing the file in the 3D Viewer Window command with a 4.0 vertical exaggeration, you can even see how the curb-and-gutter Paris Boulevard ends abruptly as it transitions to the roadside ditch template of East Road.

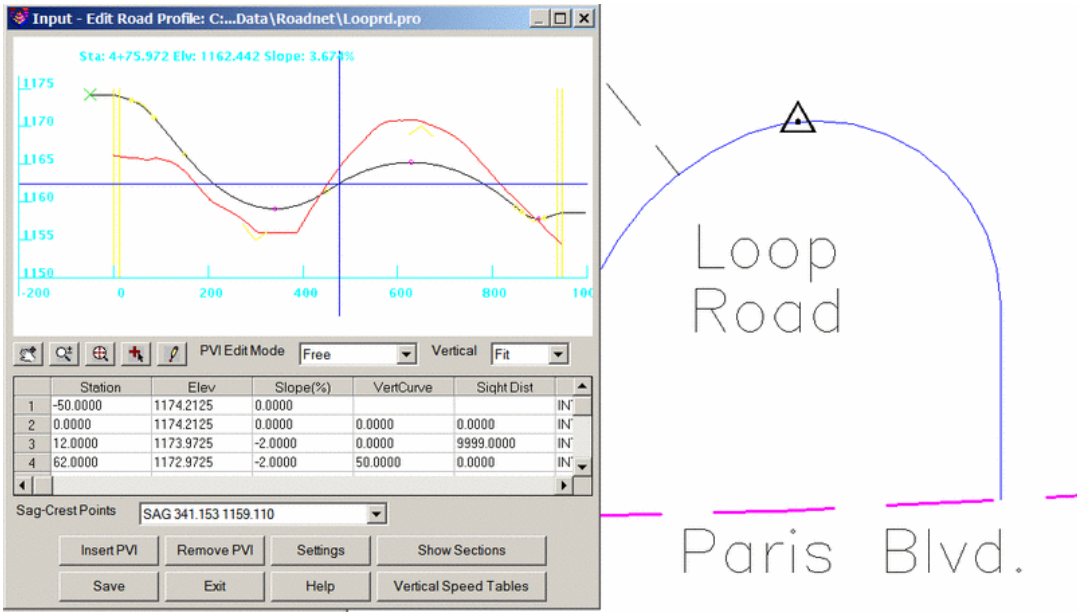


Next we can review the effect of adding Front Drive, Loop Road and West Drive into the equation. For the profile design for Loop Road and other intersecting roads, it is a good practice to use Calculate Offsets under the Centerline pulldown to compute the station of the intersection on the main road. For Loop Road, those stations are approximately 21+25 and 25+29. Then use Input-Edit Profile to load the Paris Boulevard profile and enter those stations as "Check Stations" in the lower right of the dialog, to compute the design profile at those points (1174.2 and 1158.6 respectively). These profile points can then guide the design of the Loop Road profile. After roads are added to the list, new intersections are found. When you select a road for analysis, it highlights on the drawing in plan view, as shown below.



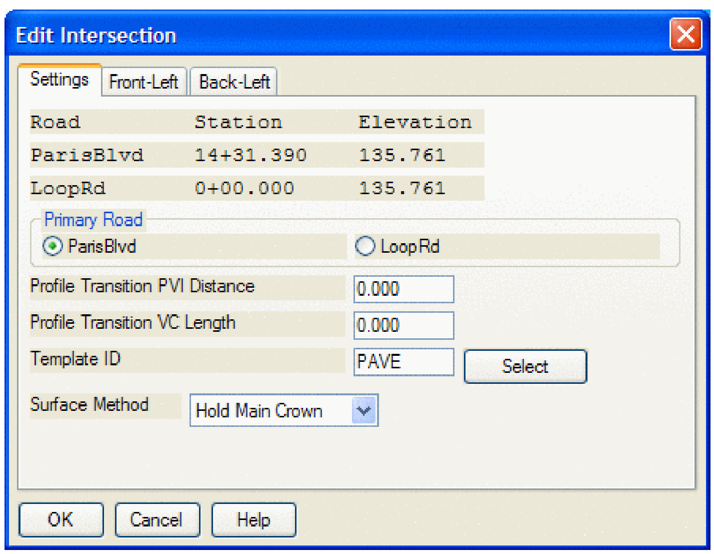
If you click Edit after selecting Loop Road as above, you have the option to change any aspect of the centerline,

profile or template file, and you can add optional files such as road width change files and superelevation files. For example, if you choose to edit the profile, the program derives the existing grade from the existing surface triangulation file specified in Settings, and you are able to design graphically and interactively as shown:



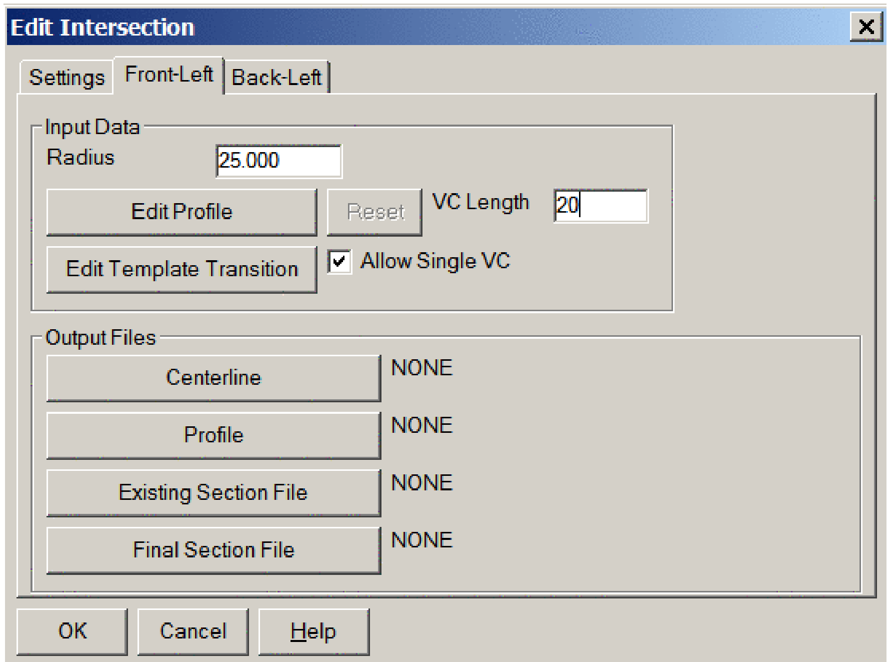
Similar editing dialogs are offered for centerline file design and for template design, in which cases you are dropped into the full editors for those commands (Input-Edit Centerline and Design Template).

You can also more closely analyze the intersections of any road. If you select the intersection at ParisBlvd and Start:LoopRd, you obtain the multi-tab dialog:

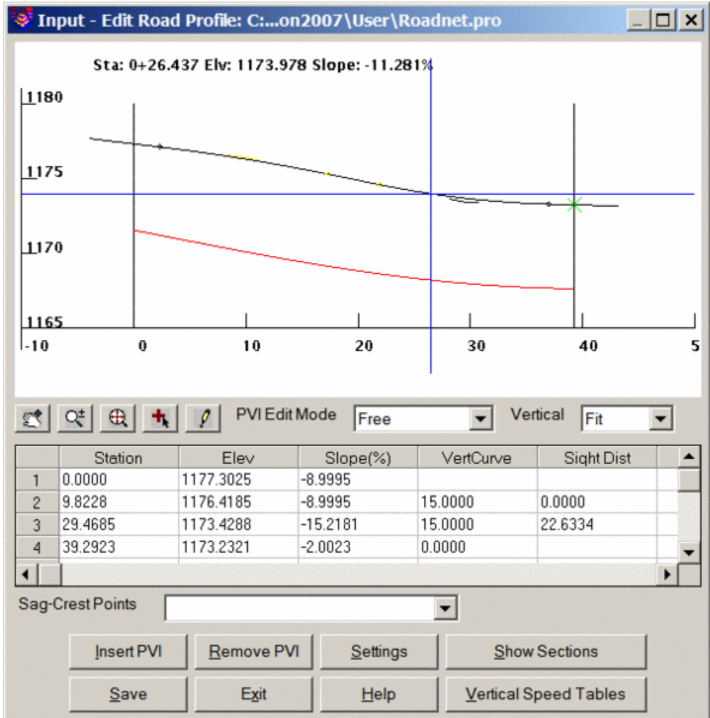


The profile transition PVI distance takes the primary road (in this case Paris Boulevard) and extends its template crown slope 50 feet, and then connects to the next PVI on the secondary road. It works on the premise that the main road's template slope must govern. It then inserts a vertical curve of specified length at the new PVI. Since 1/2 of the vertical curve is 25 feet, the actual template slope extension from the main road reaches to 25 feet from centerline. These values, inherited from the main Settings dialog, can be revised for each individual intersection.

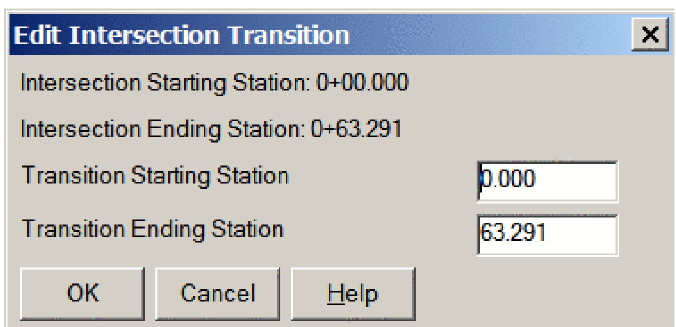
Since we do not have a crossing intersection, we only obtain a "Front-Left" tab and a "Back-Left" tab, left being the left side of the primary road (Paris) and front being the first "curve return" treatment on the outside of the loop and back being the second "curve return" treatment on the inside of the loop. If this was a crossing intersection, you would have 2 more tabs in the dialog: "Front-Right" and "Back-Right". If you click the "Front-Left" tab, you obtain this dialog:



Two edge-of-pavement alignments are intersecting, and the radius of the intersection, as defaulting from the Settings dialog, can be edited here. That covers the plan view aspect. But these edge-of-pavement points have their own profile as projected out from the governing road profile along the template. You can actually specify a vertical curve length to transition these profiles, and you can click Edit Profile to review these "curve returns" further. Clicking Edit Profile brings you to still another dialog:



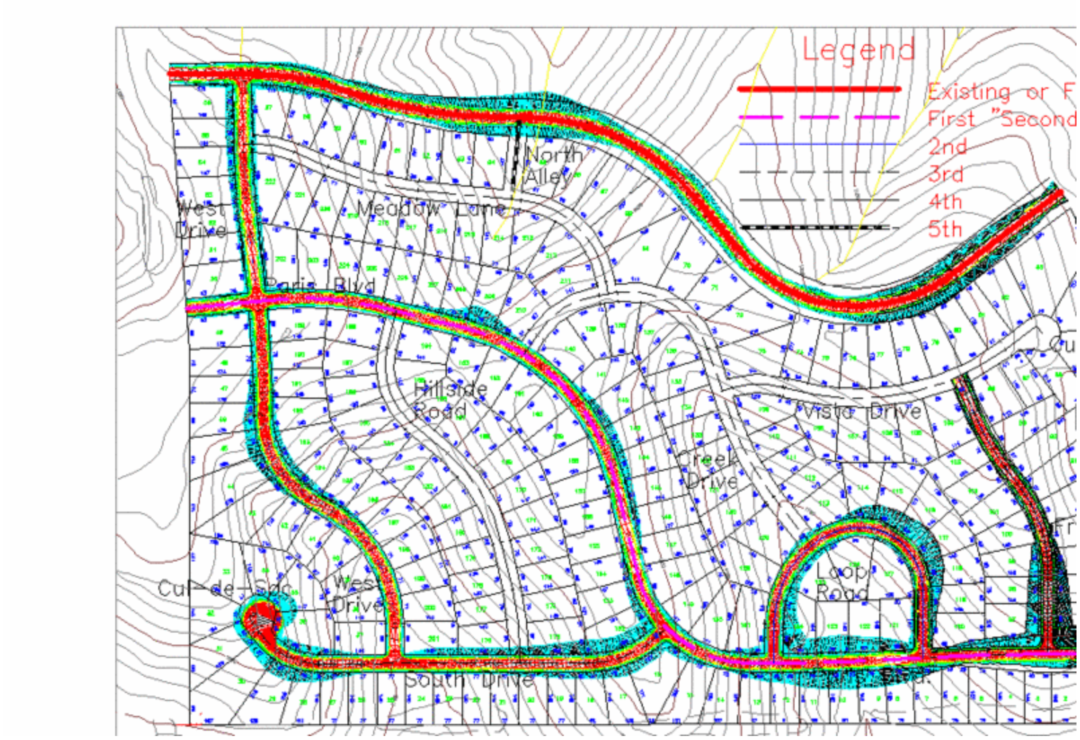
By selecting Edit Template Transition, you can shorten or lengthen the transition zone by entering new starting and ending transition stations.



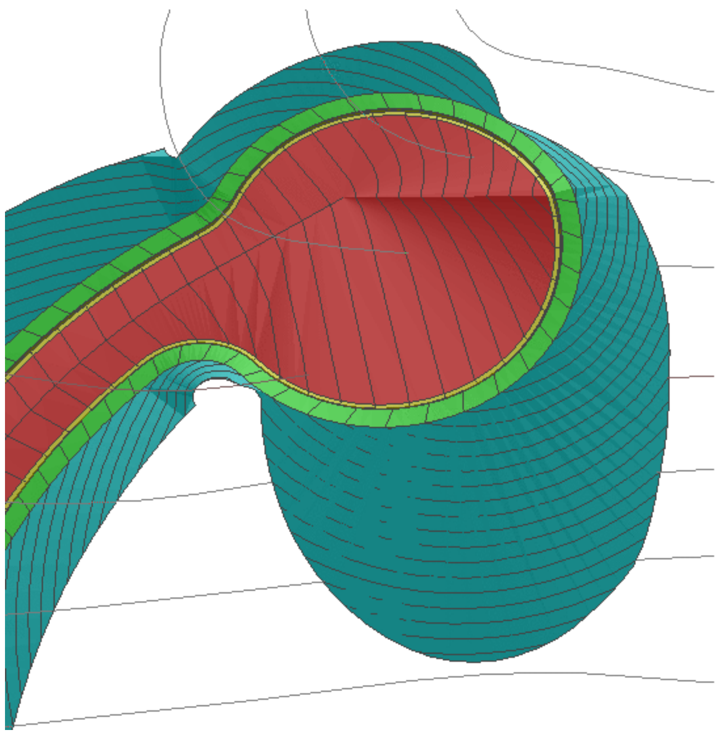
And finally, for any portion of the intersection transition, you can choose to output the centerline and profile for the shoulder pivot points, and the existing and final sections as centered on the pivot points.

Completing West Drive, Front Drive and South Drive leads to the following plan view and 3D view. Note that West Drive's profile should pay attention to the existing profile positions where it intersects Paris Boulevard and also North Road, or else excessive transition effects may occur. Using the Calculate Offsets command and just inspecting the intersect, you can note that it is found at station 194 of Paris Blvd and station 1125 of West Drive. At station 194, using Check Stations within Input-Edit Profile, the elevation is 1224.92. So the goal should be to make West Drive hit near 1224.92 at its station 1125. Similarly, Front Drive should closely match the elevation of Paris Boulevard as it starts and goes north, and South Drive is subject to both West and Paris. Clicking Add within the Cul-de-Sac portion of the docked dialog enables you to specify at cul-de-sac at the end of South Drive.

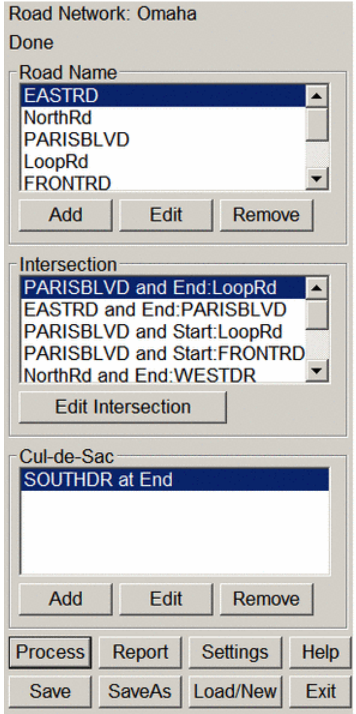
Clicking Process now produces the following:



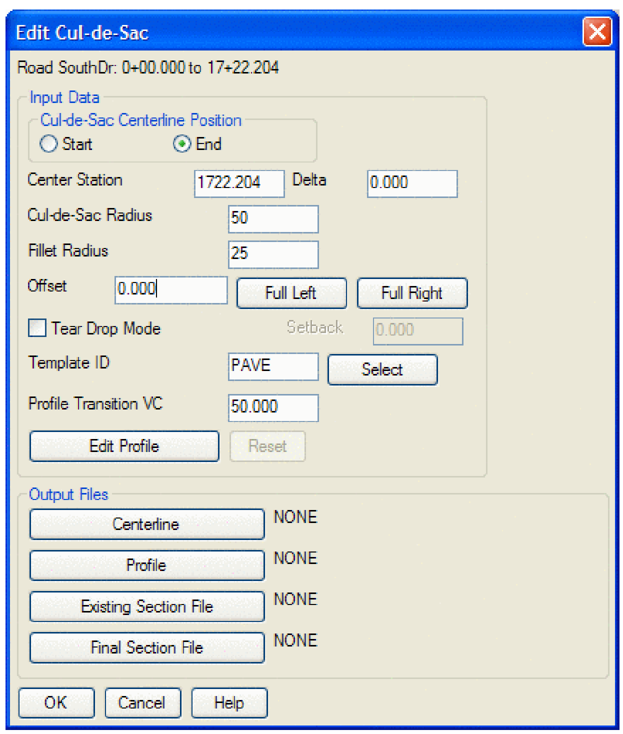
A close-up view of the cul-de-sac, in 3D, reveals the detail of the design, showing a raised "fold" due to no vertical curve transition at the projected high point at the back of the cul-de-sac:



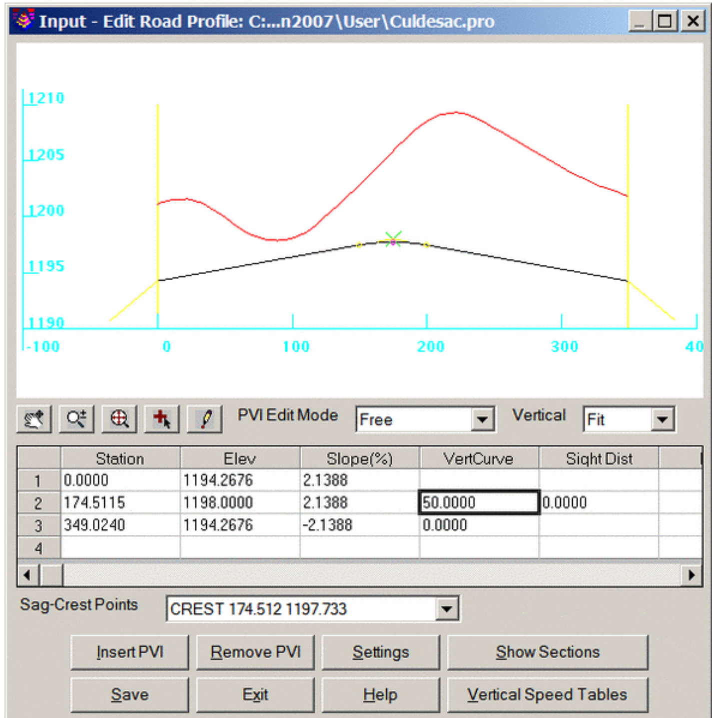
This dimple effect can easily be eliminated by lowering the elevation of the "PVI" at the projected intersect point in the back of the cul-de-sac, and by adding a vertical curve transition of, say 50'. This is done by highlighting the South Drive Cul-de-Sac and clicking Edit.



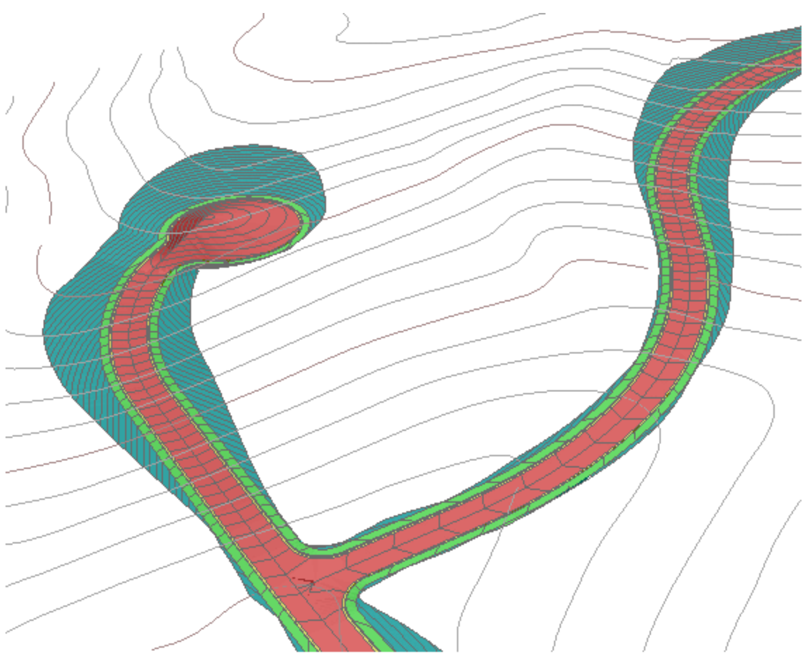
Note that now, with so many named roads and intersections, a scroll bar has materialized in the upper portions of the dialog. Clicking Edit on the selected SouthDr at End cul-de-sac leads to this dialog:



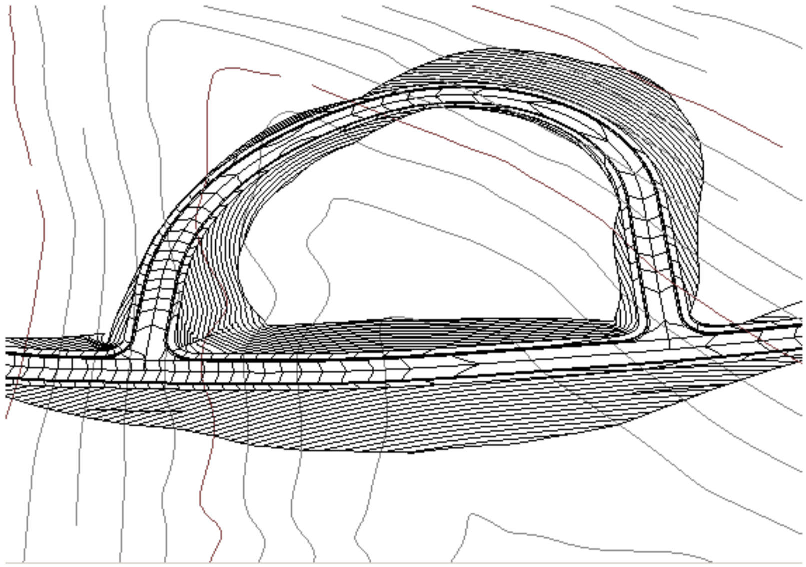
The first thing we do is change the Profile Transition VC from 0.0 to 50.0, as shown. Then we need to click Edit Profile to lower the profile at the back of the cul-de-sac. This profile refers to the edge-of-pavement grade.



Now, after clicking Process, the cul-de-sac has a better design:



With Road Network, you can directly input all roads and the program will sort out primary and secondary intersection priorities on its own, while providing you the option to edit these assignments within the Edit options within Intersections. The resulting DTM surfaces can then be studied for water flow using the hydrology features of Carlson Civil, cross sections and profiles can be plotted, and total road-related earthwork calculated.



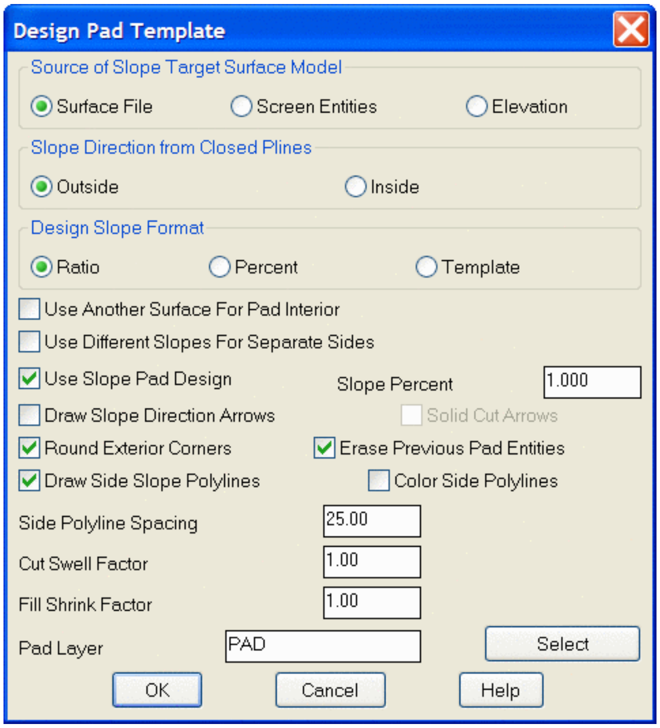
Pulldown Menu Location: Roads
Keyboard Commands: roadnet
Prerequisite: Profile file and template file
File Names: \lsp\eworkd.lsp, \lsp\eworks.arx, scadewrk.dcl

Surface Menu 12

Design Pad Template

This command creates design slopes from a perimeter polyline at specified cut/fill slopes to reach existing ground. This routine can be used to design building pads, pits, roads, ditches, stockpiles, etc. The design is drawn as 3D polylines for the cut/fill slopes and for the daylight perimeter where the design meets existing ground.

Before beginning this routine, you must have drawn the polyline representing the outside edge of the feature to model. The edge is drawn as a polyline which can be either a 2D or 3D closed or open polyline. For a 2D polyline, the program will prompt for an elevation for the pad perimeter. With a 3D polyline, the pad perimeter is set to the elevations of the 3D polyline. For an open polyline, the program will prompt for the side for the design. With a closed polyline, the program designs the slopes either outward or inward depending on the settings in the dialog.



Under **Source of Slope Target Surface Model**, choose between a Surface File (.GRD, .FLT, .TIN), Screen Entities, or a specific Elevation. If using Screen Entities, the routine internally calculates a gridded model, the limits of which are specified by screen picks. Make sure that the grid area covers the entire area for the pad including room for the cut/fill slopes.

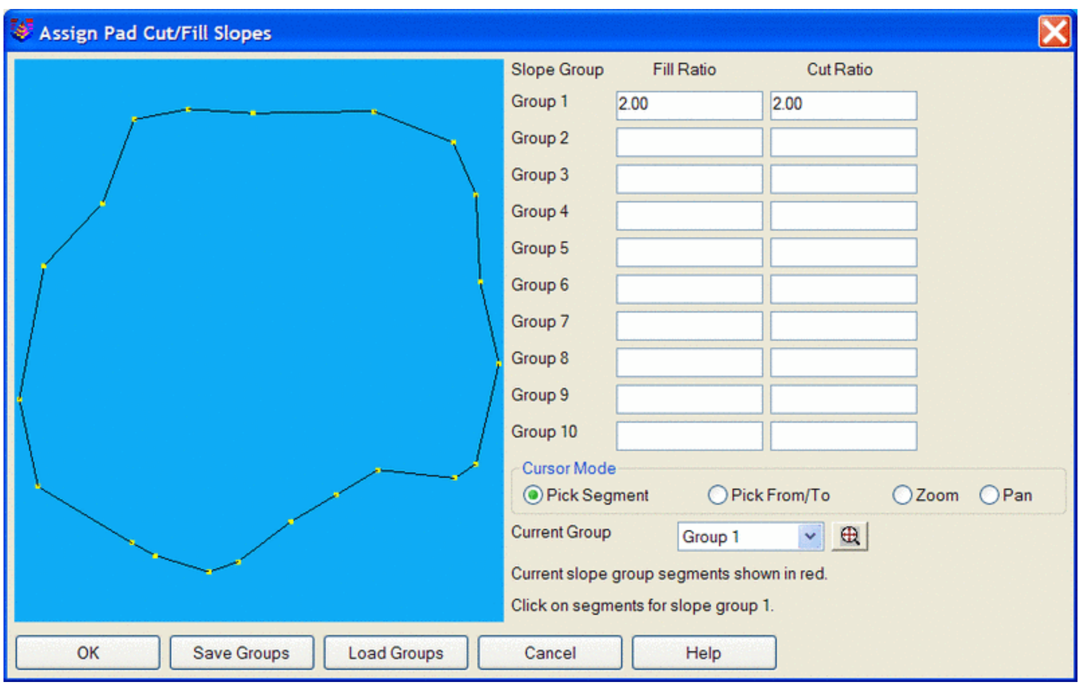
For closed pad perimeters, there is a **Slope Direction from Closed Plines** option to draw the slopes inward or outward from the perimeter. The outward method starts the slopes at the design elevation of the perimeter and projects out to intersect the existing surface. The inward method projects the slopes inside to reach the grid surface or a set elevation. Outward sloping would be used for such things as building pads, parking lots, etc. where the interior remains as a defined surface. Inward sloping would be used for such things as the top edge of an excavated pit or pond where the interior side slopes project downward at the specified slopes until reaching the original ground surface.

Under **Design Slope Format**, choose between *Ratio*, *Percent*, or *Template*. The use of a Template allows for complex

slopes to be applied, and is also an alternative approach to road design. The template (.TPL) file is created in the *Design Template* routine in the Roads menu. When using a template, the pad perimeter represents the centerline. One way to create the pad perimeter for the template is to use the *Profile to 3D Polyline* command which converts a 2D centerline to a 3D polyline using a design profile. With a template, the program uses not only the cut and fill slopes from the template file but also draws all the template grade points such as edge of road, curb and ditch. The subgrade, superelevation and template transition options of the template file are not used in this command. These options are only applied in the *Process Road Design* command. The grade points are drawn as 3D polylines parallel with the centerline. Cross section 3D polylines that include the grade points are also drawn at the specified interval.

Use Another Surface for Pad Interior will bring up a prompt for another Surface file (.GRD, .FLT, .TIN) to use for the design surface within the starting pad perimeter. Otherwise the program will model the pad interior by straight interpolation from the starting pad perimeter elevations. For example, if a building pad has a starting pad perimeter at a set elevation and the pad is supposed to be flat, then this option is not needed. This option is needed in a case where you are designing a pit and the starting pad perimeter is a 3D polyline that follows an undulating pit bottom surface. The pad design will model the pit side slopes. In order to model the undulating bottom of the pit, you need the Use Another Surface for Pad Interior option to select a surface that models the pit bottom.

Use Different Slopes for Separate Sides allows you to specify different slopes for different sides of your pad polyline. If this is toggled ON, the Assign Pad Cut/Fill Slopes dialog is invoked, where you can create multiple Slope Groups along the Pad Template polyline and set the Cut and Fill design ratios for each.



Use Slope Pad Design allows you to set a cross slope amount for the top of the pad. You will be prompted to screen pick two points that designate the slope direction. For automatic balancing of cut/fill quantities, you will be prompted to find the optimal slope and slope direction.

Draw Slope Direction Arrows draws an arrow on the outslopes that points in the downhill direction. Arrows on fill slopes are drawn as solid filled.

Solid Cut Arrows allows you to choose between drawing the cut arrows as solid filled or as wire frame.

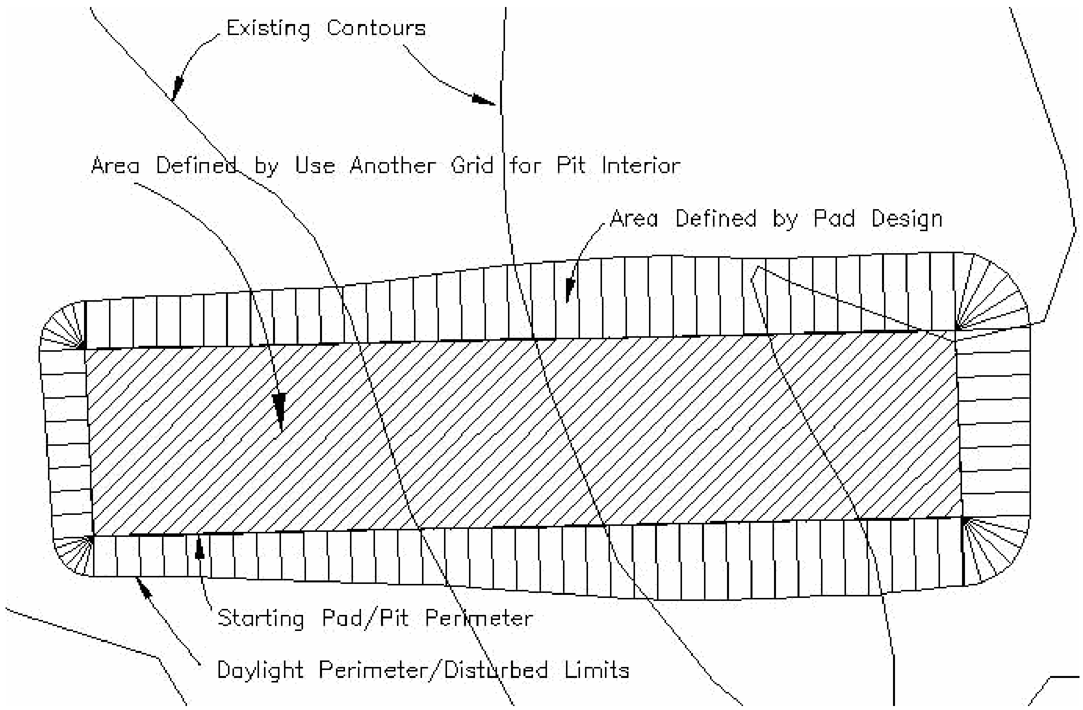
Round Exterior Corners holds the outslopes around the corners. Otherwise the side outslopes stay straight until

they meet at the corners as shown in the figure.

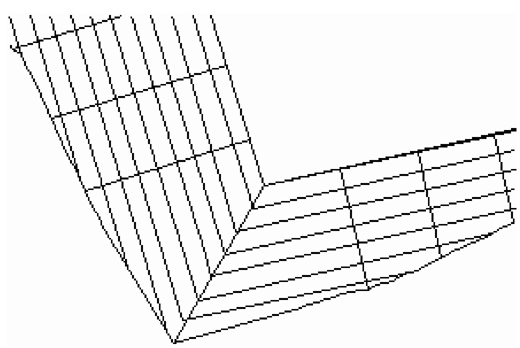
Erase Previous Pad Entities erases drawing geometry created with this command previously.

When **Draw Side Slope Polylines** is ON, Design Pad Template will draw 3D polylines perpendicular to the pad perimeter from the pad to the catch point.

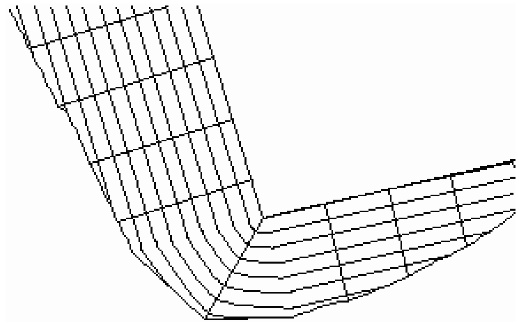
Color Side Polylines assigns different colors to Cut and Fill Side Polylines to make them easier to distinguish.



Example of pit design for option of Use Another Grid for Pad Interior



Pad corner without round corners option



Pad corner with round corners option

Side Polyline Spacing specifies the interval at which to draw the Side Slope Polylines. Besides at the interval, side slope polylines are also drawn at grid corners.

Cut volume is multiplied by the **Cut Swell Factor** in the final volume report.

Fill volume is multiplied by the **Fill Shrink Factor** in the final volume report.

You must specify the **Pad Layer Name** that the pad 3D polylines will be drawn on.

There is an option to calculate volumes for the pad design. The volumes are calculated by comparing the existing surface with the pad design. The inclusion perimeter for the volume calculation is the daylight perimeter polyline which represents the limits of disturbed area. The existing surface model is defined by the existing surface file (.GRD, .FLT, .TIN) or screen entities selected at the beginning of the command. The pad design surface is calculated by making a surface from the pad 3D polylines including the starting pad perimeter, the side polylines and the daylight perimeter.

Besides calculating the volumes in the *Design Pad Template* routine, you can also calculate the volumes with the *Two Surface Volumes* command, or the *Volumes by Triangulation* command. Two Surface Volumes works with two grid files, Volumes by Triangulation works with two TIN files. The design surface for Two Surface Volumes can be the final output surface from Design Pad or you can create a design surface with *Make 3D Grid File* using the 3D polylines created in *Design Pad*. You could also create a TIN surface of the design surface using *Triangulate and Contour*. Some of the reasons to use either the Two Surface Volumes command or the Volumes by Triangulation command are that these volume routines have more output options (cut/fill color maps, etc.) and you can check the volumes by plotting or contouring the surface files. Also, you can combine several pads and other final surfaces by running *Make 3D Grid File* or *Triangulate and Contour* and then use these volume commands to calculate the overall site volumes.

The design is drawn as 3D polylines and the earthwork volumes are calculated. Before ending, the program allows you to adjust the design by changing the pad elevation, slopes and offset. The program can find the cut/fill balance by automatically adjusting the pad elevation. If adjustments are specified, the pad polylines are redrawn and the volumes recalculated. At the end, there is an option to trim existing contours inside the disturbed limits of the pad. Then there is an option to draw contours on the pad. If contouring is selected, a dialog lets you set the contouring options. Usually you should specify a new contour layer and turn off smoothing.

A few Key things to note:

1. If the Source of Slope Target Surface Model is set to a Surface File, and the surface file used is a grid file, then the surface produced from the designed pad will be a grid surface and a grid file (.GRD).
2. If the Source of Slope Target Surface Model is set to a Surface File, and the surface file used is a TIN file, then the surface produced from the designed pad will be a triangulated surface and a TIN file (.TIN).

3. If the Surface used as a Target Surface is listed in the Surface Manager, the prompt seen in the Design Pad Template command is whether or not to Update the Surface, which is the Target Surface, so if you say "Yes," your Existing Ground Surface will now essentially contain the designed pad. So if you want to maintain an unedited version of Existing Ground, you may want to start with a copy of the Existing Ground Surface.
4. If the Surface used as a Target Surface is not listed in the Surface Manager, the prompt seen in the *Design Pad Template* command is whether or not to create a new surface of the combined surfaces.
5. If you respond "Yes" to the prompt about whether to contour the designed pad, the contouring dialog box has an option of whether to write the designed pad as a new surface, which will be only the area within the limits of the new design, not the entire Target Surface and design pad surface combined.

Prompts

First you are presented with the Design Pad Template dialog box.

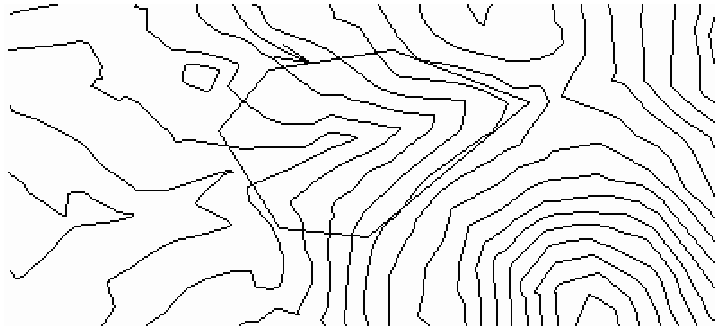
If the Source of Slope Target Surface Model is set to a Surface File, you will first be asked to:
Pick the top of pad polyline: *select perimeter polyline*
Then the Select Slope Target Surface dialog box is presented. Choose the Slope Target Surface file, pick Open. You then proceed to enter the slope parameters of the pad...

If the Source of Slope Target Surface Model is set to a Screen Entities, you will first be asked to:

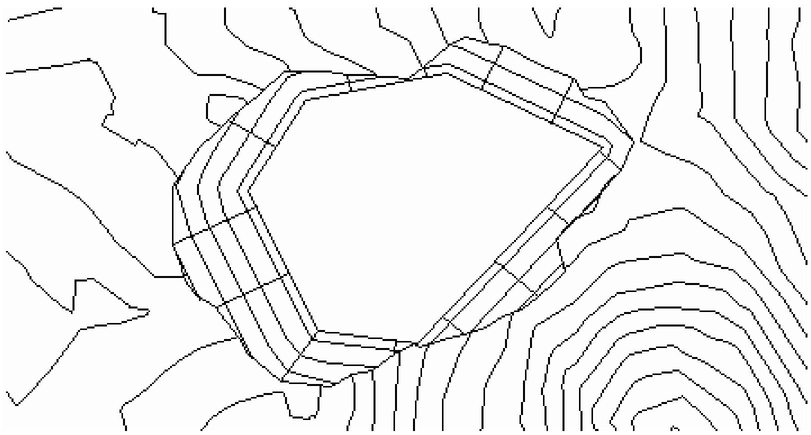
Pick Lower Left limit of pad disturbed area: *pick lower left* These prompts appear for the Screen Entities surface model method.
Pick Upper Right limit of pad disturbed area: *pick upper right* Be sure to pick these limits well beyond the area of the top of pad polyline in order to make room for the outslopes.
Make Grid File Dialog After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

Then,
Pick the top of pad polyline: *select perimeter polyline*
Then proceed to enter the slope parameters of the pad...

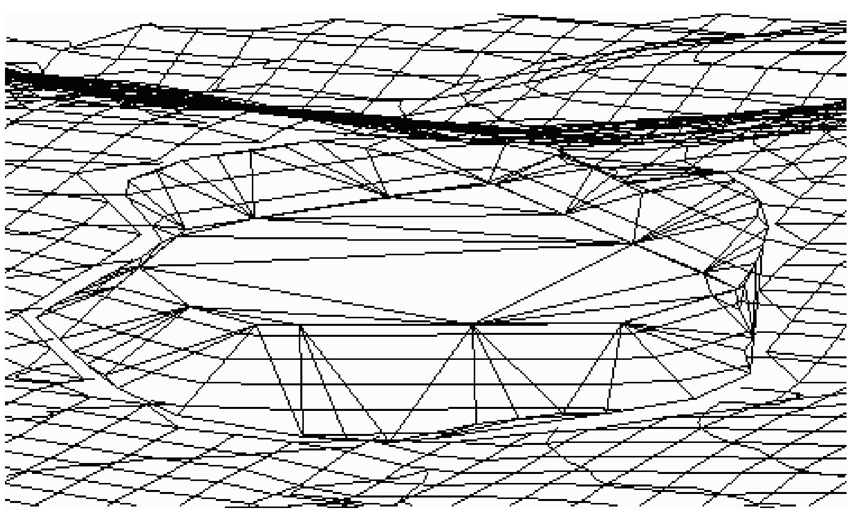
Enter the fill outslope ratio <2.0>: *2.5*
Enter the cut outslope ratio <2.0>: *2.5* After entering outslopes slope ratios, a range of elevations along the pad top will be noted.
Enter the pad elevation <29.54>: *39*
Calculate earthwork volumes (<Yes>/No)? *press Enter*
Report Viewer Reports cut/fill volume.
Adjust parameters and redesign pond (Yes/<No>)? *press Enter*
Write final surface to grid file (Yes/<No>)? *press Enter* This option will output a grid file using the elevations of the pad within the disturbed area polyline and using the original ground surface everywhere else.
Trim existing contours inside pad perimeter (Yes/<No>)? *press Enter*
Contour the pad (<Yes>/No)? *press Enter*



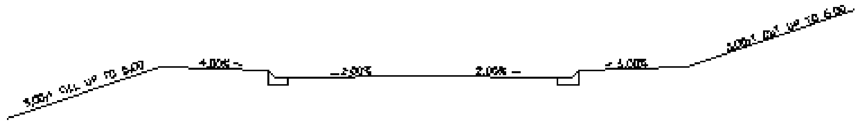
Existing contours with top of pad perimeter polyline



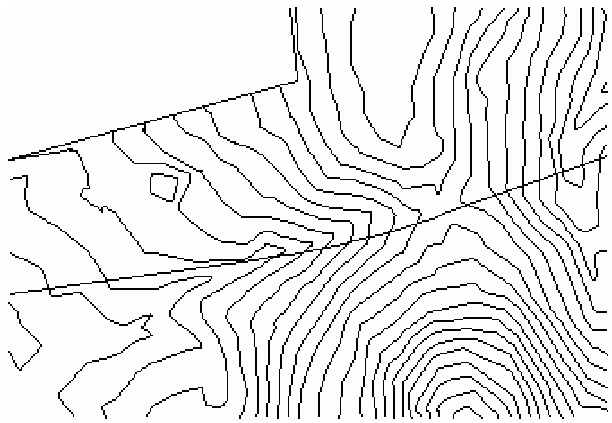
Pad template with contours



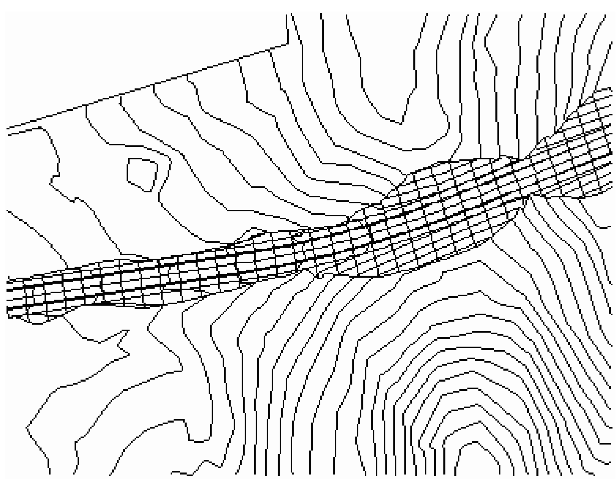
3D view of pad with DTM of surface and triangulation faces of pad



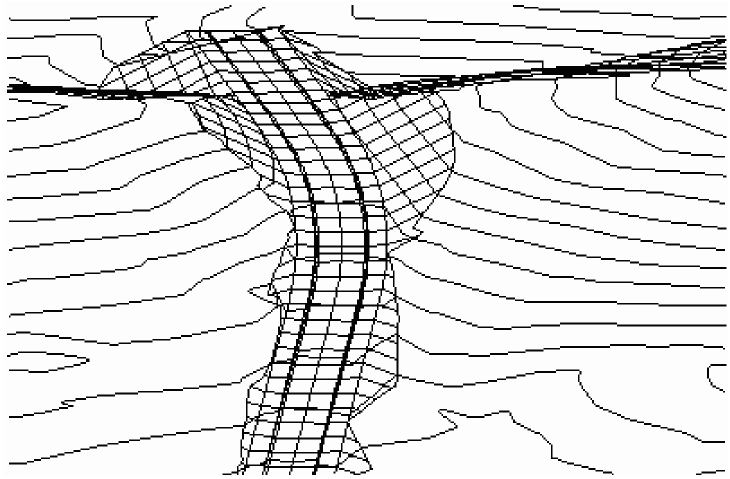
Template to apply in Design Pad Template



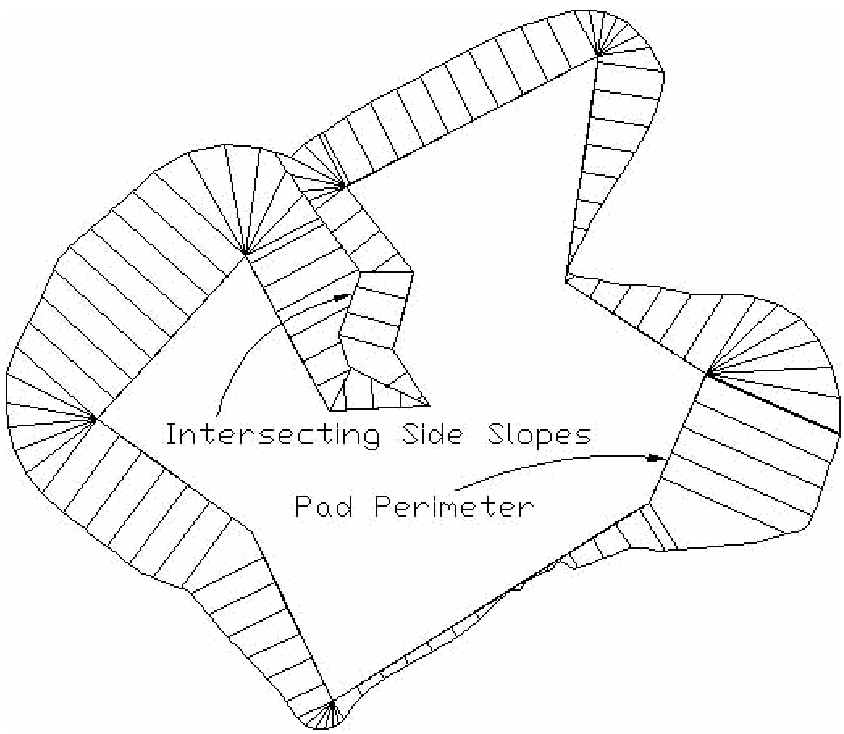
Existing surface with 3D polyline centerline



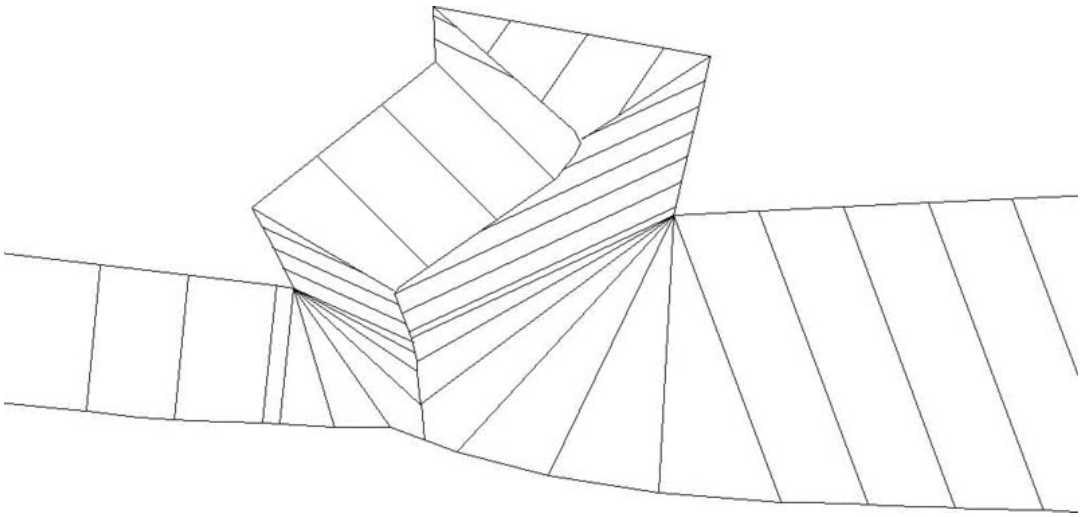
Result of Design Pad Template showing template grade polylines,
cross section polylines, cut/fill slopes, and final contours



Viewpoint 3D view of Design Pad Template



Design Pad Template can also handle self-intersecting side slopes



Viewpoint 3D view of intersecting side slopes

Pulldown Menu Location: Surface
Keyboard Command: pad
Prerequisite: A pad perimeter polyline and surface entities or a surface file for an intercept target.
File Names: \lsp\flatpond.lsp, \lsp\pond.arx

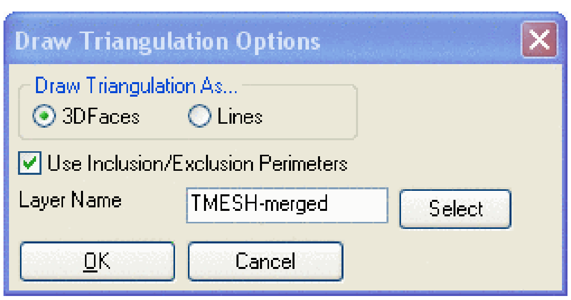
Draw Triangular Surface

This command draws a triangulation (.flt or .tin) file as either 3D LINES or 3DFACEs. Since 3DFACE entities can be shaded within the *3D Viewer Window* or *3D Surface FlyOver,* or with the AutoCAD *3D Orbit* command, this is an excellent tool for visual surface inspection. 3D Lines cannot be shaded.

Triangulation (.flt or .tin) files can be created by *Triangulate & Contour*.

Prompts

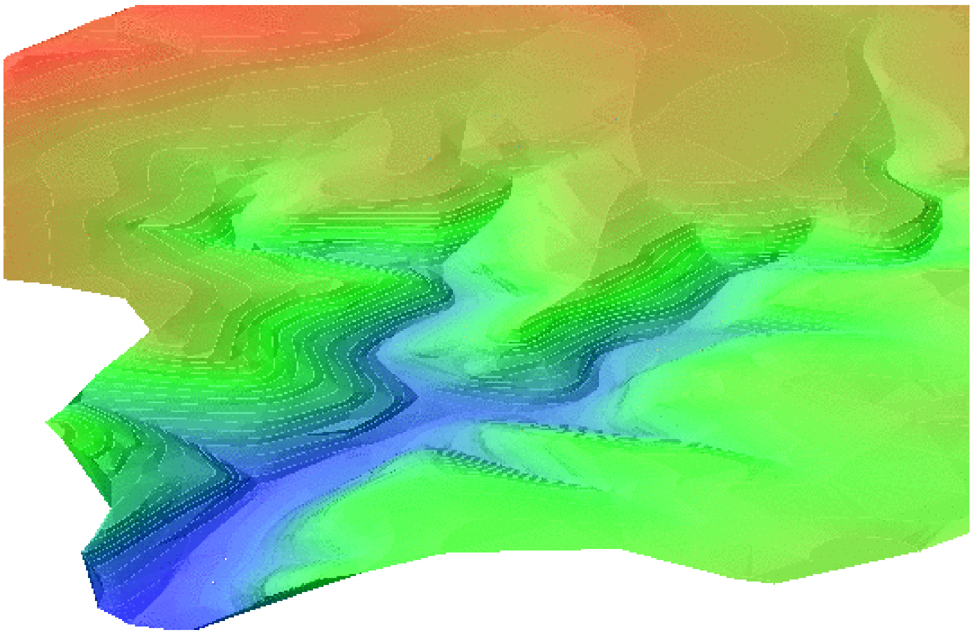
Select TMESH File to Draw
Choose a triangulation (.flt or .tin) file from the file selection dialog. You are then prompted for options:



If using Inclusion/Exclusion Perimeters, you will be prompted to select them as the routine executes.

Loading edges...

Loaded 198 points and 234 edges

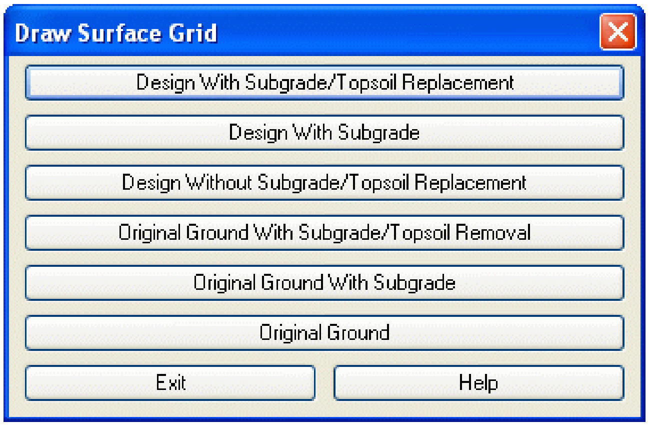


This Triangulation mesh was drawn as 3DFaces with the Draw Triangular Mesh command, and then colorized by elevation within 3D Viewer Window

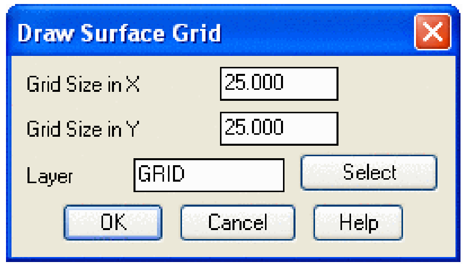
Pulldown Menu Location: Surface >> Draw Surface
Keyboard Command: drawtri
Prerequisite: A triangulation (.flt or .tin) file
File Name: \lsp\contour4.arx

Draw Surface As Grid

This command writes a grid file (.grd) from an existing triangulation file (.flt or .tin) in the current drawing.



After selecting the triangulation file to convert, you are prompted for the X, Y grid interval and the Layer name.

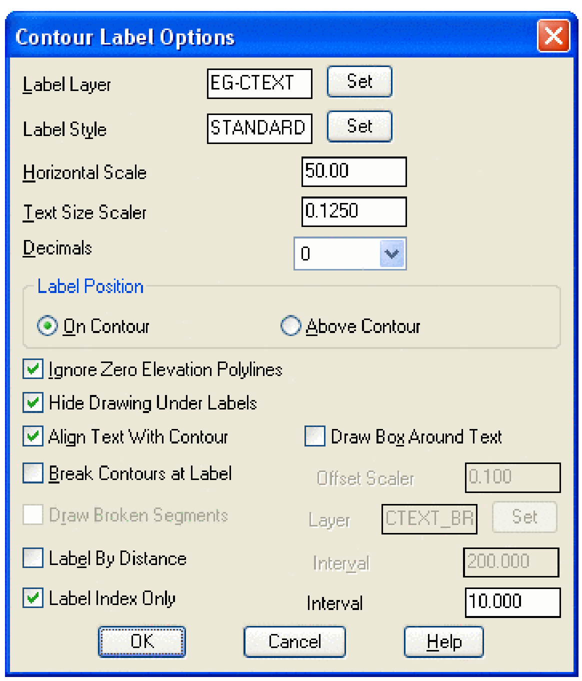


Prerequisite: a existing or design surface
Keyboard Command: draw_surface_grd

Contour Elevation Label

This command can be used to simultaneously create elevation labels on a group or groups of contour polylines at elevation. To place the labels, pick two points crossing the contour polylines at the desired label location. The program will find all the contour polylines that intersect the picked line (defined by the two picked points) and will place labels at the intersection point of each contour. A second crossing line can be initiated immediately, so multiple areas can be quickly labeled while remaining in the command. The actual "z" elevation of the contour line determines the label value.

Contour Label Options Dialog



Label Layer specifies layer name for the contour labels that will be created.

Label Style: specifies the text style to be used for labels.

Horizontal Scale is used in conjunction with the Text Size Scaler to determine unit height of the contour labels.

Text Size Scaler is a scaler that will be multiplied by the horizontal scale to set the actual text height of the labels in AutoCAD units.

Decimals sets the decimal precision for the labels to be created.

Label Position determines the label position in relation to the contour polyline.

- **On Contour** centers the label on the contour line.
- **Above Contour** places the label above the contour line. If this option is used, the options for Break Contours at Label and Draw Broken Segments become inactive.

Ignore Zero Elevation Polylines enables the routine to filter out all entities with an elevation of zero.

Hide Drawing Under Labels activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour line is still fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, yet the contour lines themselves remain contiguous. This feature will also hide other entities that are in the immediate vicinity of the contour label.

When **Align Text with Contour** is checked, contour elevation labels will be rotated to align with their respective contour lines.

When **Break Contours at Label** is checked, the contour lines will be broken and trimmed at the label location for label visibility.

When **Draw Broken Segments** is checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label By Distance places the labels by distance along the contour. The user is not prompted for screen picks of contour crossing when this option is used.

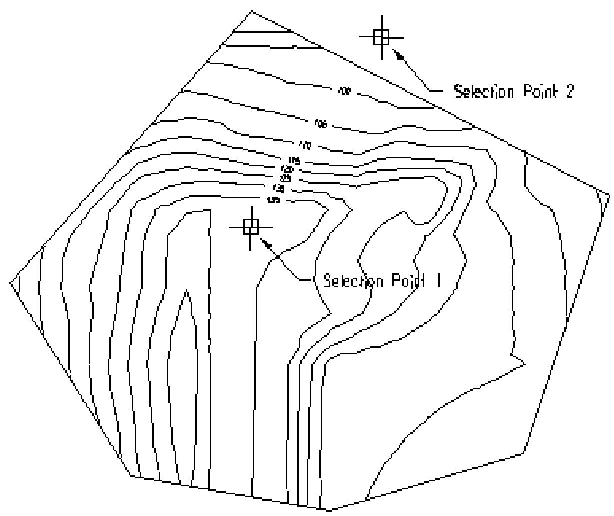
- **Interval** sets the distance interval to be used between labels on each contour.

When **Draw Box Around Text** is checked, a rectangle will be drawn around the elevation labels.

Label Index Only: When checked, only Index contours are labeled.

Prompts

Contour Label Options Dialog Opens Select the desired options and press OK.
Define a line which slices the contours at the desired label locations.
Pick 1st point: *pick a point*
Pick 2nd point: *pick a point*

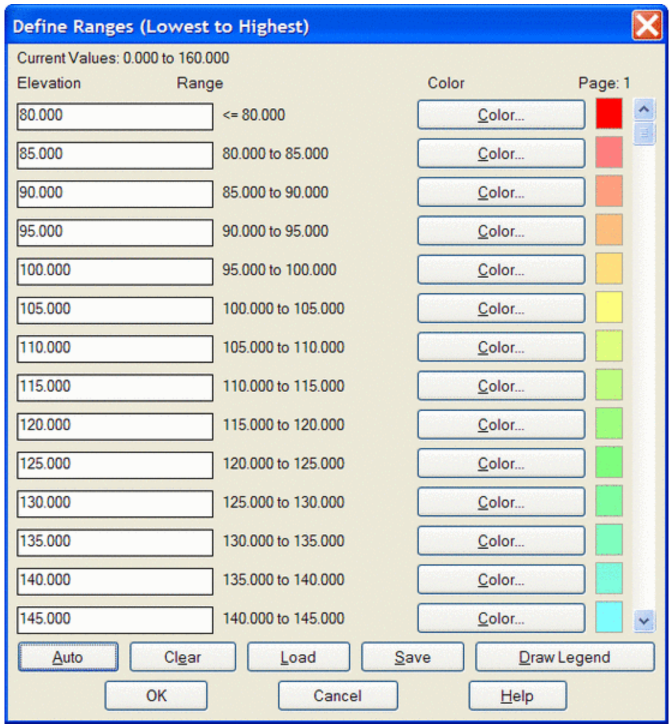


By selecting two points the contour lines that cross the line defined by the two points are labeled.

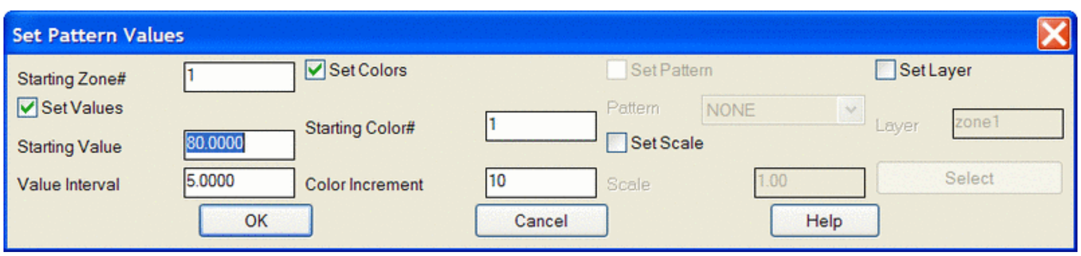
Pulldown Menu Location: Surface >> Contour Labels
Keyboard Command: gclabel
Prerequisite: polylines with elevation (contour polylines)
File Names: \lsp\gclabel.lsp, \lsp\contour4.arx

Color Contours by Elevation

This command sets the color of the selected contour polylines and text based on elevation. The color to use is defined in elevation range table.



- **Auto -** This button opens the following dialog, allowing for automatic configuration of the range of elevations and colors.



  - **Starting Zone # -** Sets the zone with which to begin the application of the settings defined in this dialog. For Instance, if the Starting Zone was set to 10, the settings definitions applied here wouldn't affect Zones 1-9, but would start at Zone 10.
  - **Set Values -** Enables the Starting Value and Value Interval fields, which allow the user to specify the starting elevation for the given zone and set the zone increment.
  - **Starting Value -** Sets the starting elevation value for the first zone.

  - **Value Interval -** Sets the elevation increment for subsequent zones.
  - **Set Colors -** Enables the Starting Color and Color Increment fields.
  - **Starting Color # -** Sets the starting color number, based on the AutoCAD standard color chart.
  - **Color Increment -** Sets the color number to increase for subsequent zones. So if the increment was set to 5, and the starting color was 60, the next color would be 65, 70, and so on.
  - **Note:** The Pattern, Scale, and Layer options do not apply to this command.

- **Clear -** Clears the all of the Elevation fields in the dialog

- **Load -** Loads previous settings from a saved .pat file
- **Save -** Saves the current setting configuration to a .pat file.

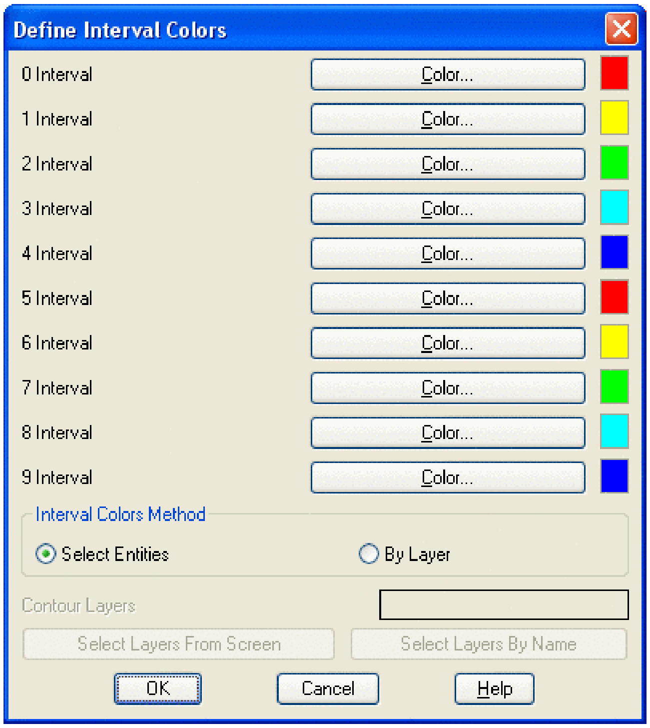
Prompts

Select polylines and text to color: *pick the entities*
Define Ranges Dialog
Pick point for color legend: *pick a point to a clear area of the drawing to place a legend or press Enter for no legend*

Pull-Down Menu Location: Surface >> Modify Contours >> Color Contours
Keyboard Command: ctrcolor
Prerequisite: Contours polylines
File Name: \lsp\contour4.arx

Color Contours by Interval

This command sets the color of the selected contour polylines based on the elevation interval values, which are essentially the number that the elevation ends with, so specific colors are assigned for elevations ending in 0, 1, 2, etc. The color assignments are defined in the Define Interval Colors dialog box.



Select Entities: User is prompted to select the contour polylines to change.
By Layer: Contour polylines are selected automatically by their layer.

Prompts

Define Interval Colors Dialog

If Select Entities is set as Interval Colors Method, *pick OK*, and you are prompted to:

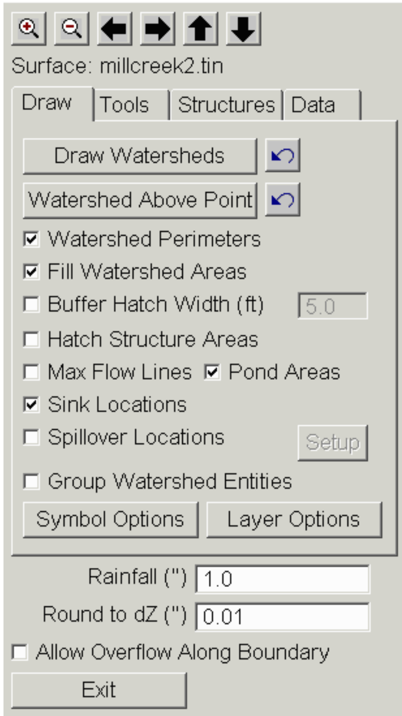
Select polylines and text to color: *pick the entities*

If By Layer is set as Interval Colors Method, set the *layers by Screen* selection or *from a list by Name*, then *pick OK*.

Pull-Down Menu Location: Surface >> Modify Contours >> Color Contours
Keyboard Command: ctrcolor2
Prerequisite: Contours polylines
File Name: \lsp\contour4.arx

Watershed Analysis

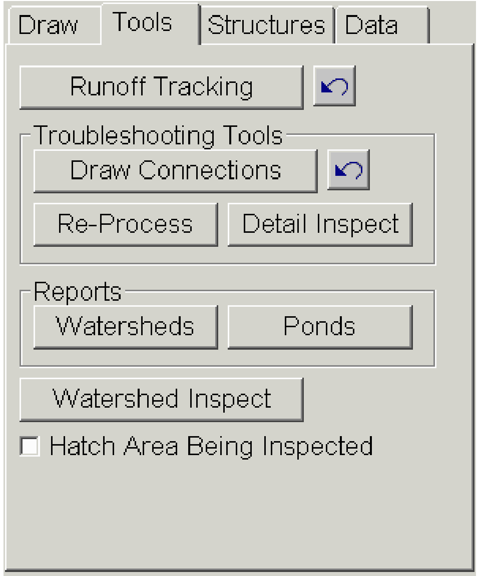
This command has a collection of tools to analyze the runoff of a surface defined by a triangulation or grid surface file. After selecting the surface file of the surface, the program docks a dialog on the left side of the drawing window. While the Watershed Analysis dialog is running, other AutoCAD and Carlson commands are not available. To zoom or pan the drawing view, use the buttons at the top of the dialog, or use the middle button of a wheel-mouse.



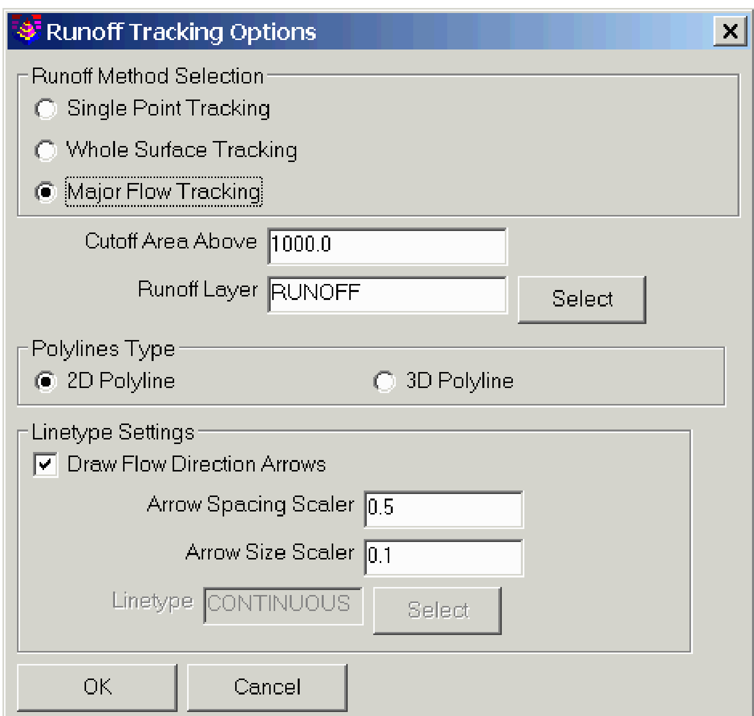
Watershed Analysis calculates the flow connections between the triangles and along the edges of the triangulation. The **Rainfall** amount is used in the processing for figuring the runoff volume to determine when the volume is enough to spillover a local depression in the surface. Besides the Rainfall amount, the runoff coefficients as defined in Define Runoff Layers are also used to calculate the runoff volumes. When the local depression is small enough the runoff will continue through. Otherwise this spot is called a sink for where the runoff stops. The **Round to dZ** is a process option that rounds the elevations of the surface model to simplify the processing. Set this value to zero for no rounding. The **Allow Overflow Along Boundary** option applies to watersheds that have runoff that hits the surface border. This option will check whether this border runoff can spillover and merge with the neighboring watersheds along the border.

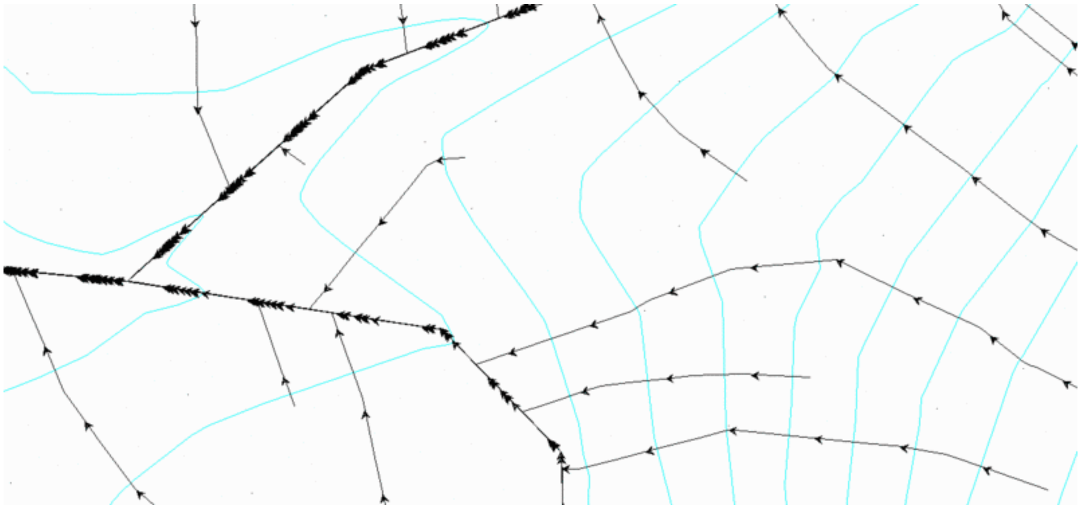
The **Draw Watersheds** function draws the watershed areas using the settings under the Draw tab. The back arrow next to the Draw Watersheds button will erase any previous Draw Watershed entities. The **Watershed Perimeters** option will draw closed polyline perimeters for each watershed area. The **Fill Watershed Areas** option will solid fill hatch each area using different colors. The **Buffer Hatch** option will hatch the perimeters of the watershed areas with the specified width instead of hatching in the entire watershed area. The **Hatch Structure Areas** option will hatch the drainage areas covered by structure inlets defined in the Structures tab. The **Sink Locations** setting draws a symbol at the low point for each drainage area. The **Pond Areas** option draws a solid fill hatch in blue for the area covered by the runoff volume of low points. In the example shown, the Fill Watershed Areas and Sink Locations options are active. The **Max Flow Lines** option draws polylines for the longest flow line within each watershed. These longest flow polylines can be used to calculate the time of concentration. The **Spillover Location** option draws symbols at low points within the watershed area that fill up with runoff and spillover on the way to the lowest (sink) location of the watershed. The Setup button allows you to specify criteria for identifying spillover points. These settings include the minimum drainage area, storage volume, drainage volume and ponding depth. These settings allow you to filter out small spillover points (ie a pothole) and only draw the significant ones. The **Group Watershed Entities** option will make AutoCAD groups for the set of entities drawn for each watershed. The **Symbol Options** and **Layer Options** buttons allow you to set the symbols and layers to use for the entities created by Watershed Analysis.

The **Watershed Above Point** function reports the watershed data of the current pointer position in real-time as the pointer is moved around. The watershed data is shown in a tooltip next to the pointer position. This data has values for the overall watershed that the position is in including the sink elevation, sink name, drainage area and average slope percent. This data also has values for the watershed above the current point including the drainage area and runoff volume. Plus this data shows the elevation and runoff coefficient at the current point. If the position is picked with the mouse, then the program draws a polyline perimeter for the drainage area above the current point.

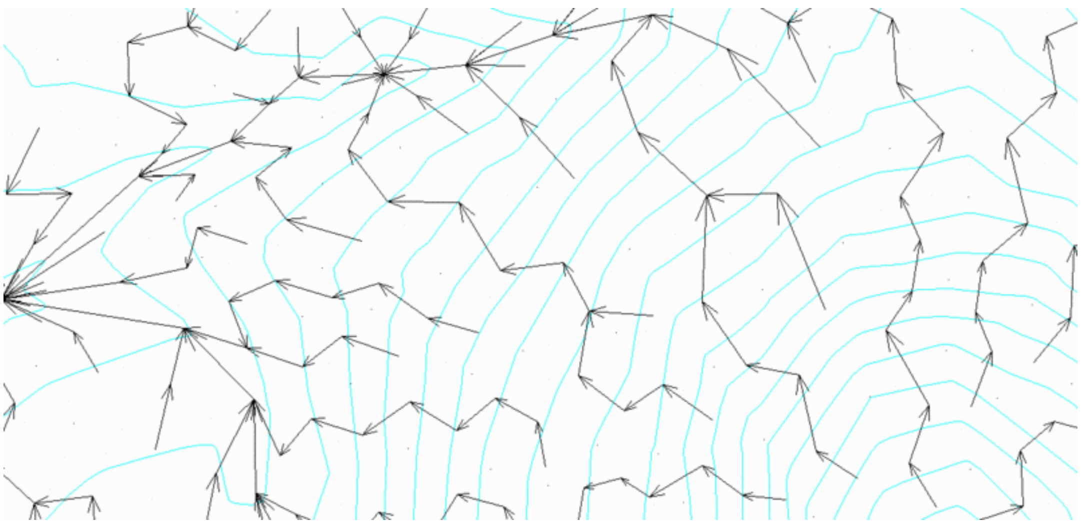


Under the Tools tab there are several analysis routines. The **Runoff Tracking** function draws flow lines that follow the surface. The **Single Point Tracking** method draws the flow lines starting from the picked high points. The **Whole Surface Tracking** method draws a flow line starting from the middle of each triangle in the triangulation. The **Major Flow Tracking** method draws starting in triangles where the drainage area coming into triangle exceeds the specified **Cutoff Area Above** value. The flow lines can be drawn as either 2D or 3D polylines. For 2D polylines, the linetype can be specified or the special linetype with flow direction arrows can be used. This special flow linetype has controls for the size and frequency of the flow arrows.





The **Draw Connections** function draws lines with arrows between the triangles for how the program has determined their flow connections.

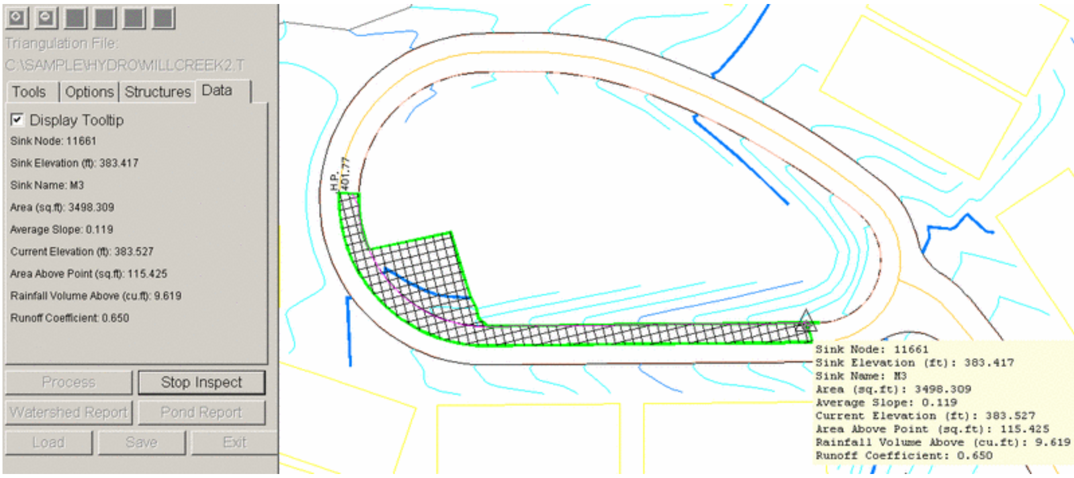


When a triangulation file is processed by Watershed Analysis, some of the flow connection data is stored into the triangulation file to speed up reprocessing. The **Re-Process** function resets this flow connection data to start the flow calculations from scratch.

The **Detail Inspect** function reports flow connection data at the pointer position in real-time as the pointer is moved. This data includes the current position triangle number, connecting flow triangle number, sink node number, watershed name, border elevation, ridge elevation, low elevation, downstream sink number, number of source triagnles, number of source nodes, current elevation and spillover elevation.

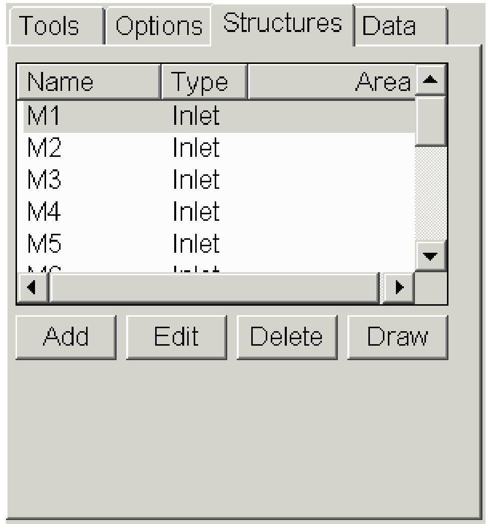
The **Watershed Inspect** function reports runoff flow data at the pointer position in real-time as the pointer is moved. The runoff data is shown in a tooltip next to the pointer and in the **Data** tab. This data has values for the overall watershed that the position is in including the sink elevation, sink name, drainage area and average slope percent. This data also has values for the watershed above the current point including the drainage

area and runoff volume. Plus this data shows the elevation and runoff coefficient at the current point. When the Hatch Area Being Inspected option is active, the watershed area for the current position is hatched during inspection.



The **Watersheds Report** function runs the report formatter to choose which of the watershed parameters to report. The **Ponds Report** function reports the position and depth of each ponding area.

Besides calculating the runoff of the triangulation surface, Watershed Analysis can also process the runoff effects from structures for inlets, storage ponds, culverts and channels. The structures in Watershed Analysis are simply for placement and watershed delineation. These structures do not have design considerations for parameters like pipe size. In the **Structure** tab, there is a list of the structures to apply with the current surface. The list shows the name, type and drainage area for each structure. The Draw function will draw symbols for each structure. The Inlet structures act as sinks in the watershed and capture all the flow that comes to the inlet point. Each inlet is defined by a single point and a name. The Storage Tank structures also act as sinks and are defined by a single point and name. The Culvert structures route the flow from the culvert inlet to the outlet. The culverts are defined by two points for the inlet and outlet and by a name. The Channel structure is the same as the Culvert except that it can have more than two points to define the flow path. The structure data can be stored to a Watershed Structure File (wst) using the **Save** button. The **Load** button can read the structure data from either a wst file or from a sewer network file (.sew).



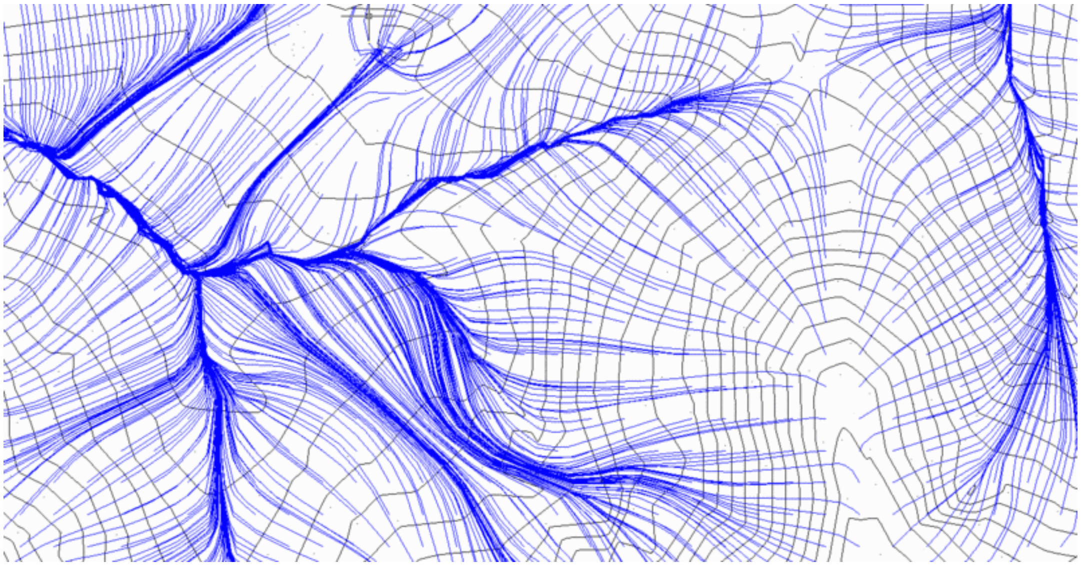
Pulldown Menu Location: Watershed
Keyboard Command: watershed
Prerequisite: Triangulation File
File Name: \lsp\cntr_grd.arx

Run Off Tracking

This command draws 3D polylines starting at user picked points downhill until they reach a local minimum or the end of the grid or TIN. In effect it simulates the path of a rain drop. The surface is modeled by a grid file as created by Make 3D Grid File or a triangulation file created by Triangulate & Contour. The program also reports the horizontal and slope distances, average slope, maximum slope, and vertical drop. These values can be used for time of concentration calculations. Runoff tracking is a convenient way to identify distinct watershed areas and is an alternative to the automated Watershed Analysis command.

Prompts

Enter the run off path layer <RUNOFF>: *press Enter*
Select Surface Model dialog box
Choose the grid file or triangulation file that models the surface. If a grid is selected, it will prompt:
Extrapolate grid to full grid size (Yes/<No>)? *Yes* If the limits of the surface data doesn't cover the entire grid area, then the values for the grid cells beyond the data limit must be extrapolated in order to compute slopes in that area. This prompt only appears if there are grid cells without values.
Local pond spillover depth <4.80>: *press Enter* This allows the runoff line to continue past flat or low points in the grid or TIN, by allowing these area to fill up with water, in essence, up to the specified depth, thus letting the runoff polyline continue on.
Draw tracking for all grid cells or pick individuals [All/<Pick>]: *press Enter* Pressing Enter leads to individual picking of runoff tracking lines, while A for All would fill draw runoff polylines starting from each grid cell or each triangulation triangle.
Pick origin of rain drop: *pick a point at the top of the run off polyline*
Pick origin of rain drop (Enter to end): *press Enter*



Pulldown Menu Location: Watershed
Keyboard Command: runoff
Prerequisite: A .grd file created by Make 3D Grid File or a .flt (TIN) file created by Triangulate & Contour.
File Name: \lsp\cntr_grd.arx

Cut/Fill Map Legend

This command will draw a Cut/Fill Map Legend on your drawing. It will display the cut/fill amount, color, and range, as seen below.

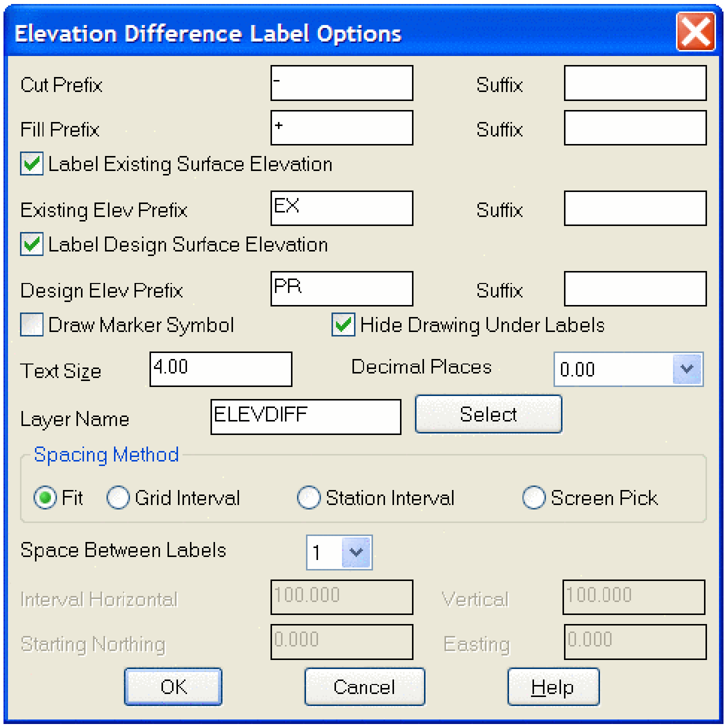
Prompts:

Select point for color legend:
Legend size <10.0>:

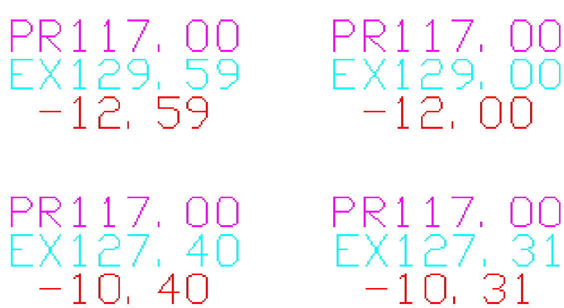
Prerequisite: Cut/Fill amounts
Keyboard Command: CF_MAP_LEGEND

Cut/Fill Labels

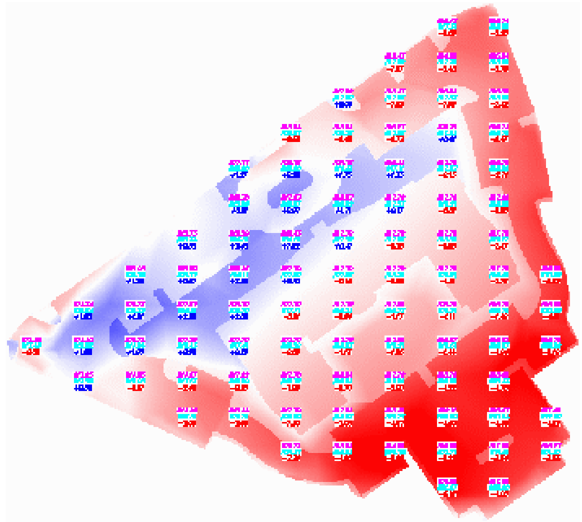
This command displays the design elevation, the existing elevation, and the amount to either cut or fill directly on the screen (See *Display Options* for information about labeling options). The following image shows the main dialog box for setting the labeling options.



The labeling created with these options looks like this:



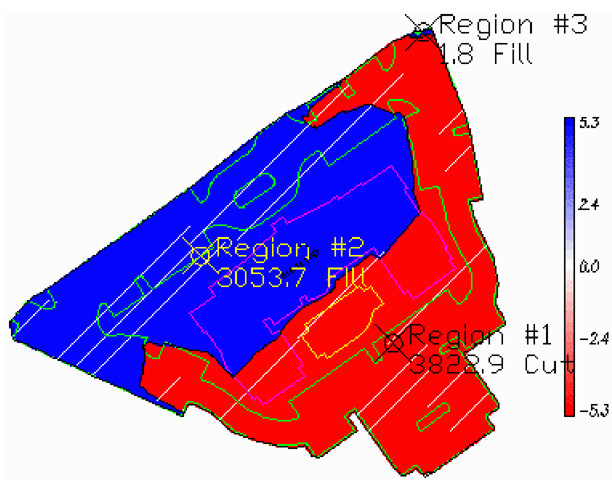
The distribution of the labels on the site looks like this:

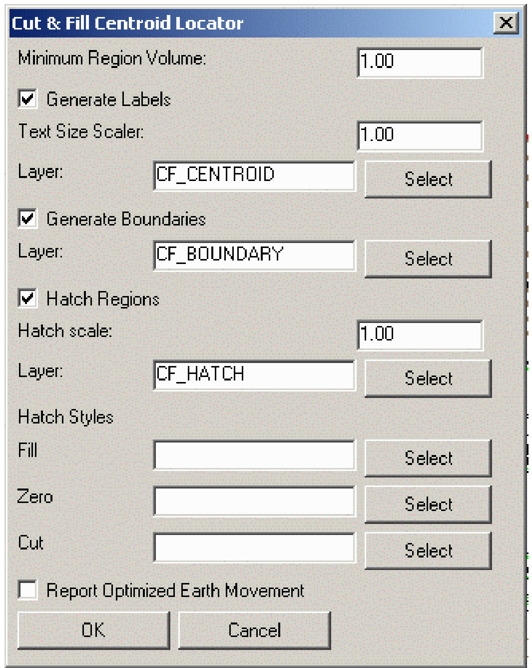


Keyboard Command: set_display_cf_txt
Prerequisite: Existing and design surfaces

Cut/Fill Centroids

This command visually shows cut/fill areas in your drawing and finds the center of mass or centroids for every cut and every fill region. Also, a report can be created to show volumes in each region.





Points are created to show the centroids locations. Also, labels can be created to display the volumes in the different regions.

```
Cut & Fill Centroid Report

Original Ground: G:\oem4\src2\work\demo1-ex.flt
Design Surface:  G:\oem4\src2\work\demo1-fn.flt

Region   Volume(C.Y.)     Northing     Easting
1        3822.9     Cut   2190032.48   6135170.29
2        3053.7     Fill  2190117.40   6134982.48
3        1.8        Fill  2190334.67   6135199.41

Earth Movement Report:
From Region   To Region   Volume(C.Y.)   Distance
1             2           3053.7         206.12
1             3           1.8            303.59
1             External    767.4
Total Internal Volume * Distance: 629972.20
Total External Volume: 767.44
```

Here is the Cut and Fill Centriod Report for the above example. It shows the volumes, the coordinates of the centroids, and the Earth Movement Report. The Earth Movement Report shows the minimal distances for moving Cut to Fill areas.

Prompts

Loading edges...
Loaded 1199 points and 3391 edges
Created 2193 triangles

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Loading edges...
Loaded 8732 points and 25651 edges
Created 16920 triangles

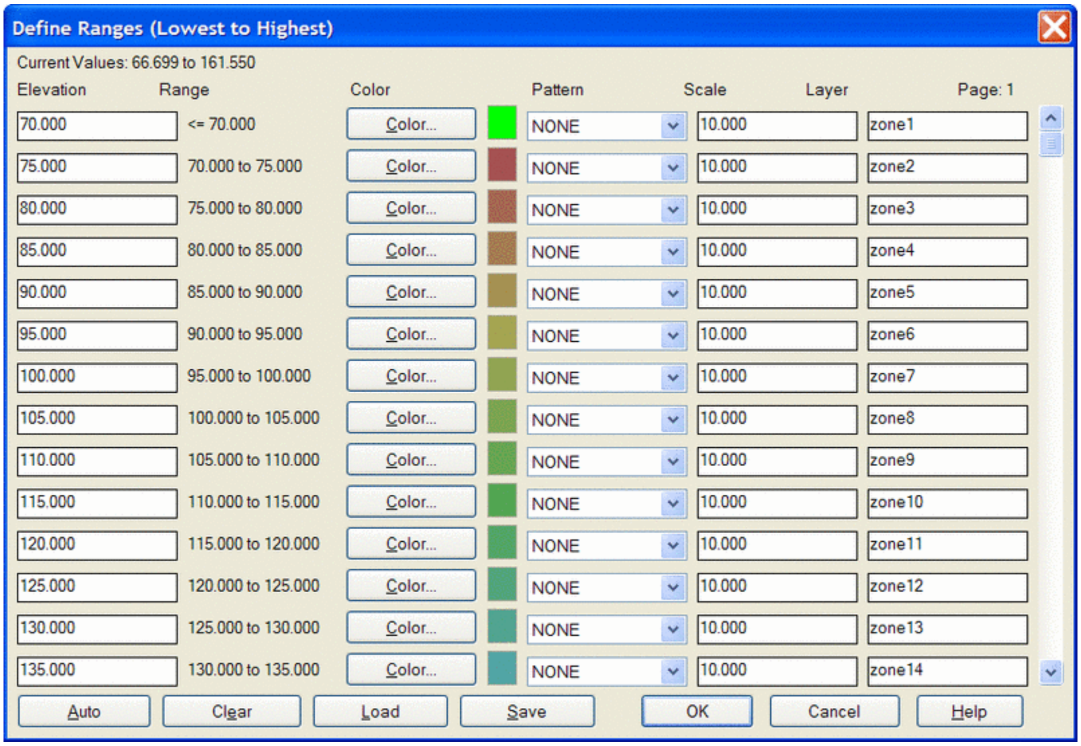
Generating report:
Region #2
3053.7 Fill
Reading the selection set ...
Joining ...
Joined 437 entities.
Region #1
3822.9 Cut
Reading the selection set ...
Joining ...
Joined 636 entities.
Region #3
1.8 Fill
Reading the selection set ...
Joining ...
Joined 49 entities.

Prerequisite: Existing and Design surfaces
Keyboard Command: tk_cutfillc

Elevation Zone Analysis

This command can be used to calculate the surface area of a surface in different elevation zone ranges and to analyze a surface by ranges or "zones" of elevation. The program requires 3D Face entities that can be generated by the *Draw 3D Grid File* command. The *Draw Triangulation Faces* option in *Triangulate & Contour* or *Draw Triangular Mesh* routine under Surface >> Draw Surfaces menu can also be used to create triangular 3D Faces. For each elevation zone, the 3D Faces can be hatched with an AutoCAD hatching pattern, solid filled with the SOLID pattern, or left empty with the NONE pattern. The 3D Faces are also placed in a separate layer for each zone.

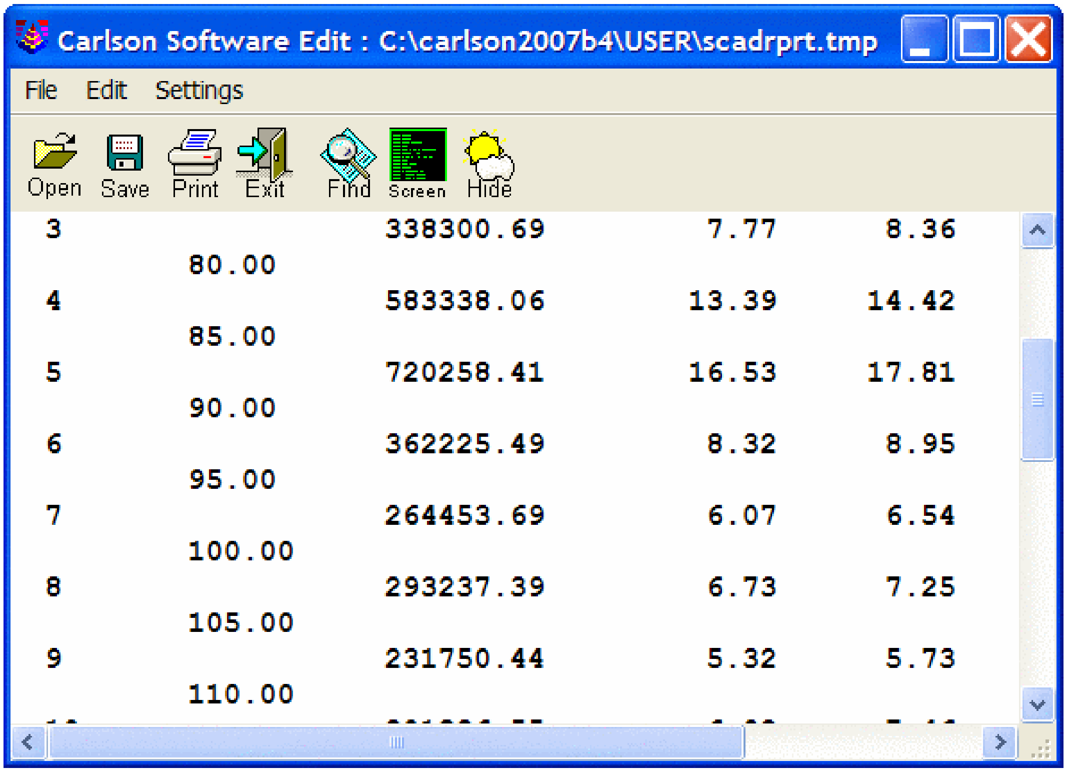
There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the area within the inclusion polyline is calculated. Areas within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the area of each selected 3D Face is used.

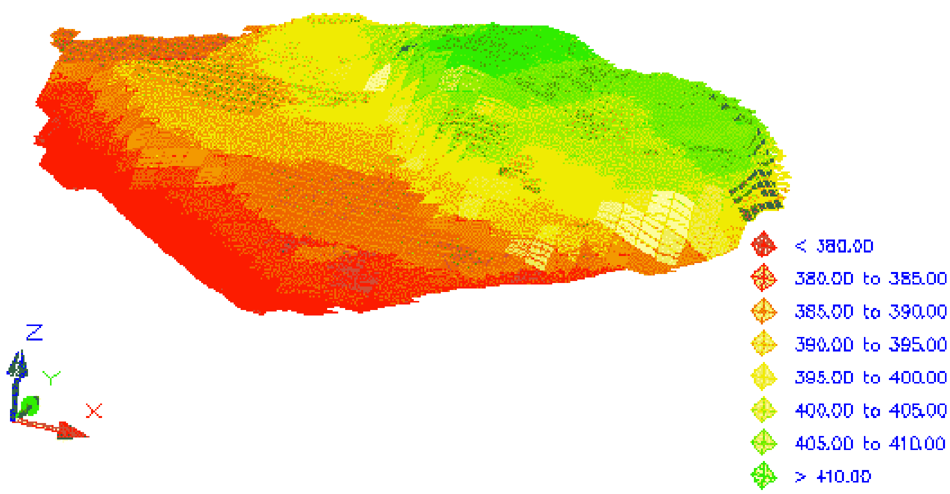


Prompts

Name of property represented by Z value <Elevation>: *press Enter*
Plot average elevation text in each grid cell (Yes/<No>): *Y*
Select 3D Faces to Analyze...
Select objects: *pick the 3D Faces to process*
Define Ranges (Lowest to Highest) Dialog
Specify the elevation ranges, colors and patterns.
Select the Inclusion perimeter polylines or ENTER for none:
Select objects: *pick a closed polyline for the limits of disturbed area*
Select objects: *press Enter*
Select the Exclusion perimeter polylines or ENTER for none:
Select objects: *press Enter*
Select point for color legend (Enter for none): *pick a point*
If a point is picked, a legend showing the color of each range is drawn. The legend is drawn aligned to the current view *UCS*. For this reason it is best to have the mesh at the *Vpoint* at which it will be plotted before executing the analysis program.

A report is also generated in the standard report viewer.





Result of Elevation Zone Analysis viewed in 3D and shaded

Pulldown Menu Location: Surface
Keyboard Command: elvzone
Prerequisite: displayed 3D Face entities.
File Names: \lsp\elanal.lsp, \lsp\contour4.arx

Slope Report

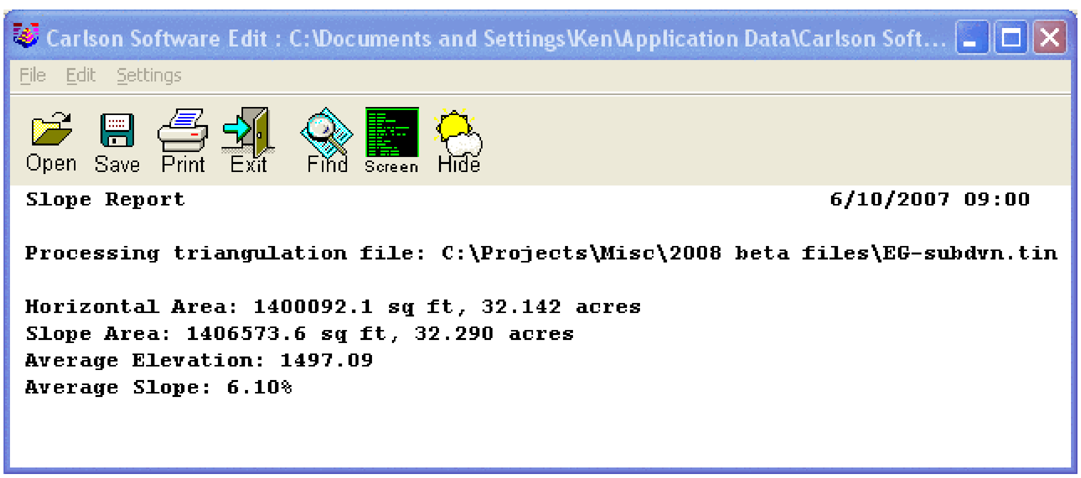
This command calculates the sloped surface area, average slope and average elevation on a site. The surface can be defined by a surface model file, (.GRD, .TIN or .FLT), or generated from 3D entities on the screen. Sloped area information is useful to compute seeding quantities for hillsides, for example.

For the screen method, the surface is modeled from the user-selected entities such as contour polylines. Besides the surface entities, a perimeter polyline is used as the inclusion area for the slope report. If the perimeter polyline is on the PERIMETER layer, the command will locate it automatically.

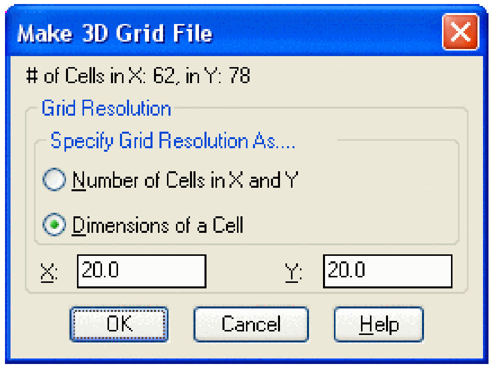
For area reports, there are options to specify inclusion and exclusion perimeters. When inclusion perimeters are specified, only the area within the inclusion perimeters is calculated. The area within exclusion perimeters is not included in the calculations. Inclusion and exclusion perimeters are represented by closed polylines and must be drawn prior to running this routine.

Prompts

For Area report using a File:
Slope report by area or two points [Area/<Points>]? *A for Area*
Source of surface model (<File>/Screen)? *F for File*
Select surface model file.
Select the Inclusion perimeter polylines or ENTER for none: *pick any inclusion polylines*
Select the Exclusion perimeter polylines or ENTER for none: *pick any exclusion polylines*
Note: If the surface model file is a grid file (.GRD), you are prompted whether to extrapolate the grid to full grid size.



For Area report by *Screen* method:
Slope report by area or two points [Area/<Points>]? *A for Area*
Source of surface model (<File>/Screen)? *S for Screen*
Ignore zero elevations (<Yes>/No)? *press Enter*
Select surface entities and perimeter.
Select objects: *pick the objects*
If no polyline is found on layer PERIMETER, you are prompted to: Select Pond/Pit perimeter polyline.
The Make 3D Grid File dialog is presented. Pick *OK*.



Select the Inclusion perimeter polylines or ENTER for none: *pick any inclusion polylines*
Select the Exclusion perimeter polylines or ENTER for none: *pick any exclusion polylines*

For *Points* method:
Slope report by area or two points [Area/<Points>]? *P for Points*
Select surface model file.
Pick first point:
Pick Second point:
The slope report is displayed on the command line for the 3D vector, projected on the surface, defined by those 2 picks.

Point 1: 5119.646,5640.322,98.979
Point 2: 4951.964,6022.419,135.546
Horiz Dist: 417.27 Slope Dist: 418.87 Elv Diff: 36.57
Slope: 8.76 Ratio: 11.41:1

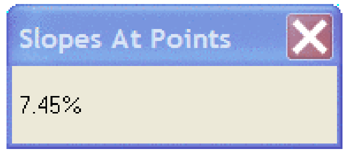
Pulldown Menu Location: Surface >> Slope Analysis
Keyboard Command: sarea
Prerequisite: A surface file or screen entities of the surface.
File Names: \lsp\sloparea.lsp, \lsp\volcalc.arx

Slope At Points

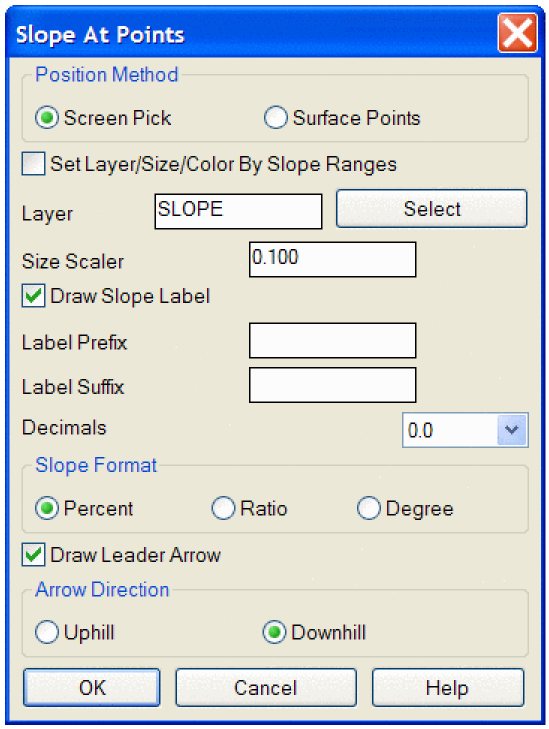
This command labels the slope percent at user Screen Picked points or Surface Points. Surface Points can work well on grid files, (.GRD), but is typically too much information for triangulated surface files (.TIN or .FLT).

The slope is computed from the surface model file (.TIN, .GRD, or .FLT).

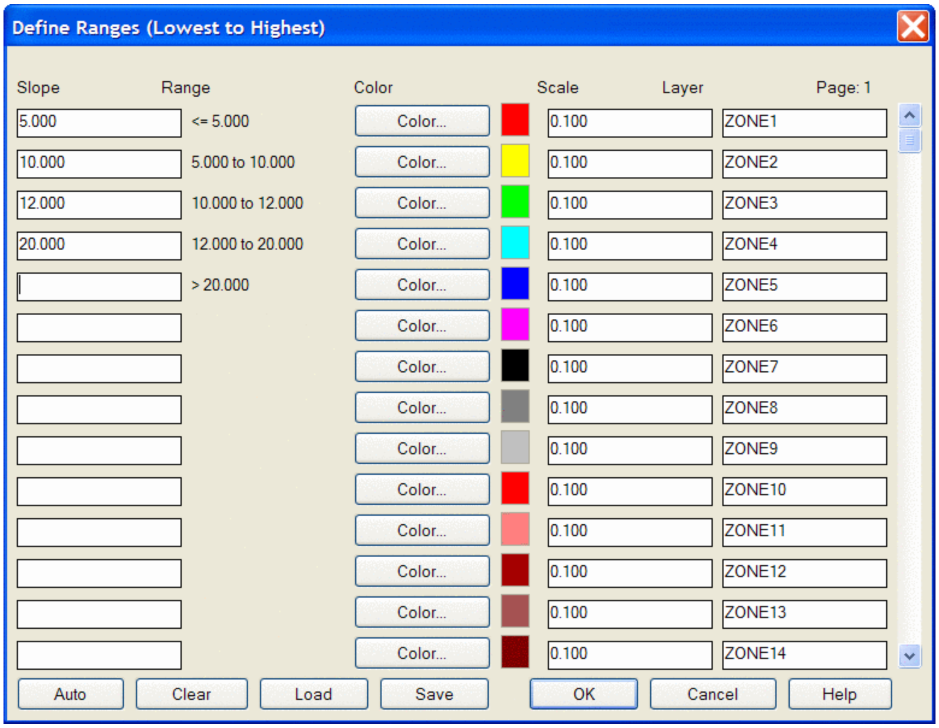
As the crosshairs are moved across the surface, the slope at the current position is displayed in a floating dialog box.



In addition to labeling the slope value at the user specified points, a Leader Arrow can be drawn in either the uphill or downhill direction. The dialog also allows you to specify Label Prefixes and/or Suffixes, Decimal Precision, and Slope Format.



The Set Layer/Size/Color By Slope Ranges option invokes the Define Ranges dialog box. Enter slope values in the first column of boxes to set the Ranges.



Prompts

Slope At Points dialog box
Adjust settings as desired. Pick OK.

Select Surface Model.

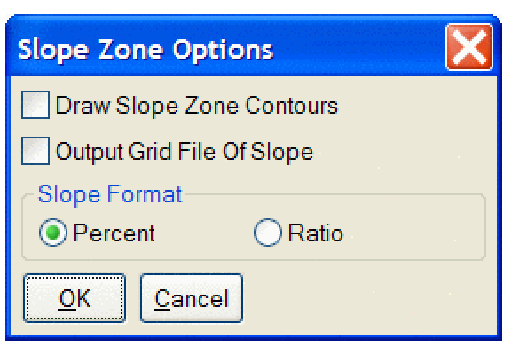
Pick Points to label slope.

Draw grid file and Slope At Point labels using Surface Points

Pulldown Menu Location: Surface >> Slope Analysis
Keyboard Command: ptslope
Prerequisite: A surface model file (.TIN, .GRD, or .FLT)
File Name: \lsp\cntr_grd.arx

Slope Zone Analysis

This command calculates the surface area of a site in different slope zone ranges. This command can use either a surface model file, (.TIN, .GRD, or .FLT), or 3D Face drawing entities, which can be generated by the *Plot 3D Grid File* command, the *Draw Triangular Mesh* command, or the Draw Triangulation Faces option of *Triangulate & Contour*. For each slope zone, the 3D Faces can be hatched with any AutoCAD hatch pattern, including the SOLID pattern, or left empty with the NONE pattern.

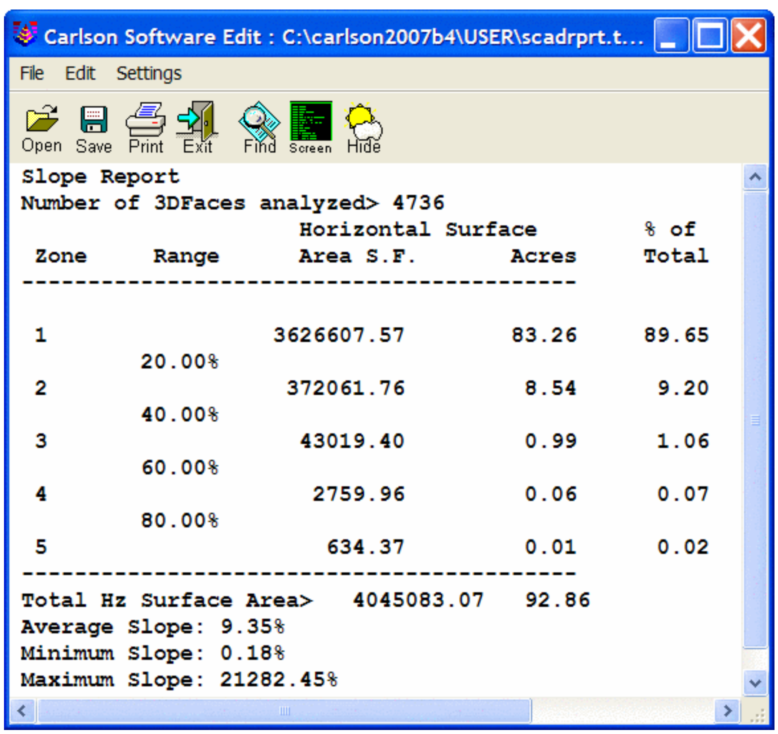


This command can also generate contours of the slope zones based on the calculated slope at each point of the 3D Faces. The slopes can vary greatly between neighboring points. When contoured directly, these slope data points produce incoherent contours. Instead this routine applies a filtering algorithm that reduces the noise. There is another option to output a grid file of the slope values.

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the slope area within the inclusion polyline is calculated. Slope area within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the slope area of each selected 3D Face is used.

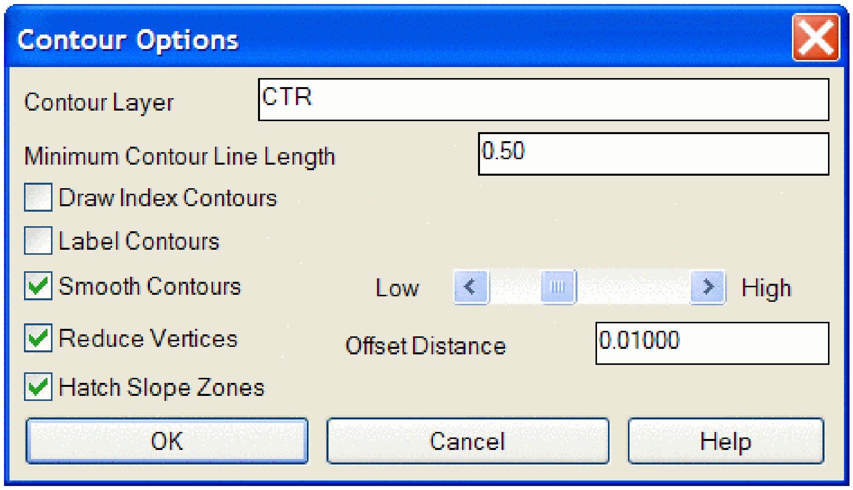
Prompts

Source of surface model: [File/<Screen>]? *F* for File
Slope Zone Options dialog box. Choose whether to Draw Slope Zone Contours, whether to Output Grid File of Slope, and Slope Format. Pick *OK*
Select surface model file.
Define Ranges dialog. Specify the slope zones, colors and patterns from lowest to highest. Pick *OK*.
Select the Inclusion perimeter polylines or ENTER for none: *select perimeter(s)* or *press Enter*
Select the Exclusion perimeter polylines or ENTER for none: *select perimeter(s)* or *press Enter*
Report is generated.



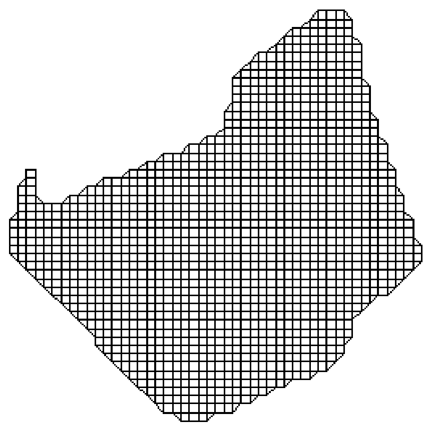

```
Slope Report
Number of 3DFaces analyzed> 4736
                    Horizontal Surface       % of
 Zone      Range        Area S.F.      Acres      Total
-----------------------------------------

 1                 3626607.57       83.26      89.65
          20.00%
 2                  372061.76        8.54       9.20
          40.00%
 3                   43019.40        0.99       1.06
          60.00%
 4                    2759.96        0.06       0.07
          80.00%
 5                     634.37        0.01       0.02
-----------------------------------------
Total Hz Surface Area>    4045083.07    92.86
Average Slope: 9.35%
Minimum Slope: 0.18%
Maximum Slope: 21282.45%
```

If you choose to draw Slope Zone Contours, the Contour Options dialog box is presented.

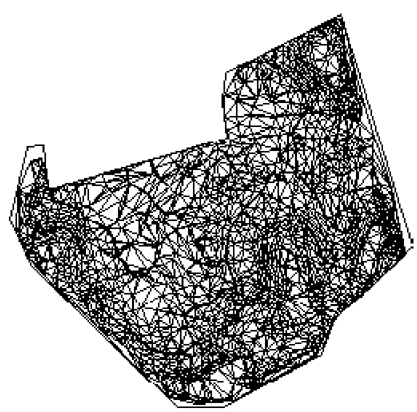


Note: If you choose to use Screen entities instead of a surface model file, you are prompted whether to:
Apply hatch patterns to grid cells [Yes/<No>]? and
Freeze grid layer after processing [Yes/<No>]?

Surface contours

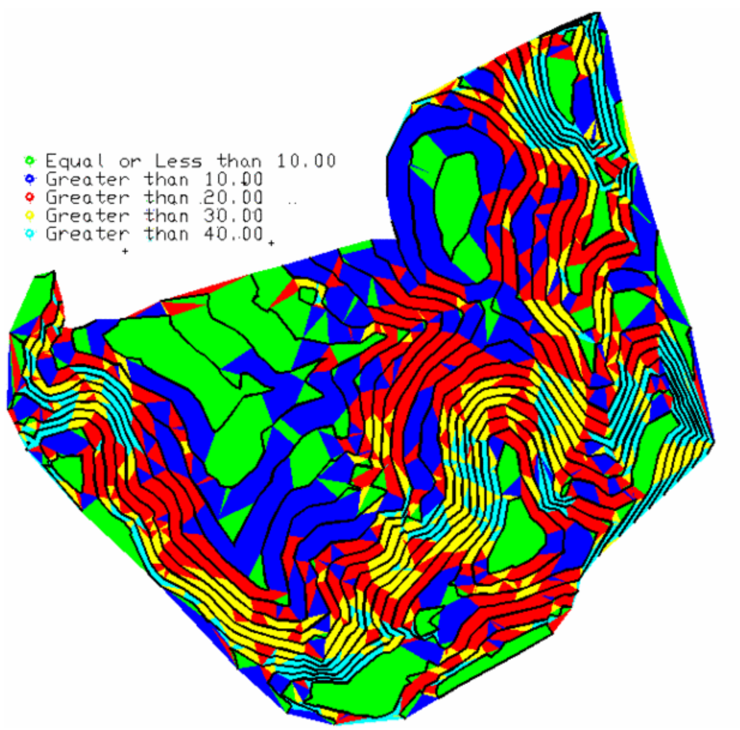


3D Faces from a grid surface model

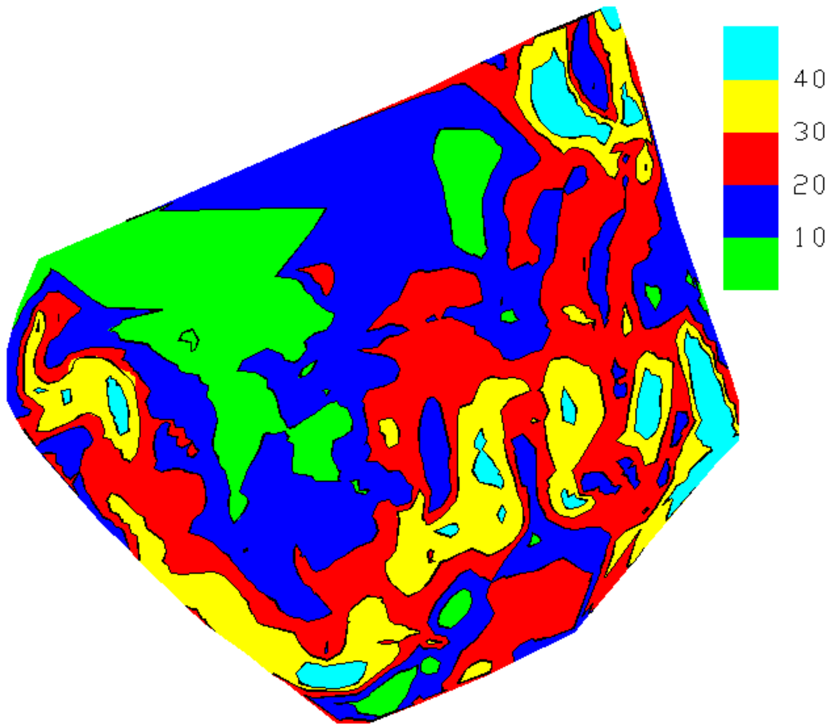


3D Faces created by *Triangulate & Contour* with the Draw Triangulation Faces option

Slope zone contours



Slope zones that follow the surface contours using the triangulation 3D Faces



Hatched slope zone contours created from the grid 3D Faces

Pulldown Menu Location: Surface >> Slope Analysis
Keyboard Command: szone
Prerequisite: Surface model file (.TIN, .GRD, or .FLT), or 3D Faces entities
File Names: \lsp\elanal.lsp, \lsp\contour4.arx

Points Menu 13

Point Defaults

This command sets Carlson point options.

Descriptions: Specify whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.

Elevations: Specify whether you are prompted for a point elevations when creating points and whether the point elevations are labeled in the point block.

Locate on Real Z Axis: When checked, points are located at their actual elevation, otherwise points will be located zero elevation.

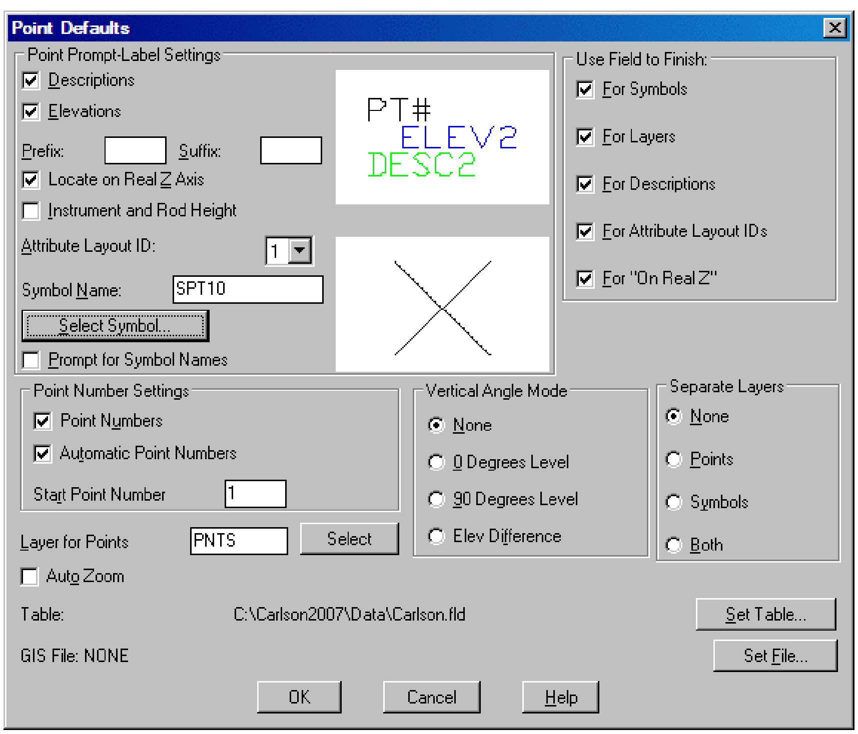
Attribute Layout ID: Controls the location of the point number, elevation and description. These attribute layouts are defined in AutoCAD drawings that are stored in the Carlson SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing.

Symbol Name: Enter the default symbol name to use. You may also pick the Select Symbol button to select a symbol from the symbol library.

Prompt for Symbol Names: When checked, you will be prompted for each symbol name instead of using the default symbol.

Point Numbers: When this toggle is OFF, no point number will be created and no points will be stored in the coordinate (.CRD) file.

Automatic Point Numbers: When this toggle is OFF, commands that locate a point will prompt for a point number. Otherwise, point numbers are numbered sequentially. If the Start Point Number field is set to 0, no point will be plotted. An exception to this is when you use the *Draw-Locate Points* command and use the Range option, then a point entity is plotted.



The following table illustrates the effects of elevation settings:

```
          Elevations Yes  Real Z No
Picked Point  Labels point, Prompts for elevation, uses 0 for z coordinate
Point Number  Labels point, No Prompt, uses 0 for z coordinate

          Elevations Yes  Real Z Yes
Picked Point  Labels point, Prompts for elevation for z coordinate
Point Number  Labels point, No Prompt, uses z coordinate from file

          Elevations No   Real Z No
Picked Point  No Label, No Prompt, uses 0 for z coordinate
Point Number  No Label, No Prompt, uses 0 for z coordinate

          Elevations No   Real Z Yes
Picked Point  Labels point, No Prompt, uses z coordinate of picked point
Point Number  Labels point, No Prompt, uses z coordinate from file
```

Start Point Number: Specify the next point number to use.

Vertical Angle Mode: Specify how Carlson should prompt you for vertical angles. None means no prompt. Applies to creating points with commands such as *Traverse.* The vertical angle is used to calculate the point elevation.

Separate Layers: Specify settings for point attribute layers.

None: The point symbol, point number, elevation and description use the layer names PNTMARK, PNTNO, PNTELEV and PNTDESC.

Points: The point number, elevation and description layers are composed by concatenating the point layer and the string NO, ELEV, and DESC respectively. For example, if the point layer is UTIL then the attribute layers will be UTILNO, UTILELEV and UTILDESC.

Symbols: The point symbol layer is composed by concatenating the point layer and the string MARK. For example, if the point layer is UTIL then the symbol layer will be UTILMARK.

Both: The point symbol, point number, elevation and description layers are composed by concatenating the point layer and the string MARK, NO, ELEV, and DESC respectively. For example, if the point layer is UTIL then the symbol/attribute layers will be UTILMARK, UTILNO, UTILELEV and UTILDESC.

Layer for Points: Specify the layer name for Carlson points.

Auto Zoom: When checked, AutoCAD will perform a Zoom—Center around new points to keep the display centered around current working area. This only applies during commands such as Traverse. This setting is also available in *Configure* under General Settings where it is called Auto Zoom Center for New Points.

Use Field to Finish For: Allows you to use the code definitions from Field to Finish for the Point Symbols, Layers, Descriptions, Attribute Layout IDs and whether to locate the point on the "Real Z" and whether to Separate Attribute Layers when creating new points. For example, when creating a point with description "EP", Carlson would look up "EP" in the Field to Finish table and will use the field code definitions to establish the point instead of the definitions defined in Point Defaults.

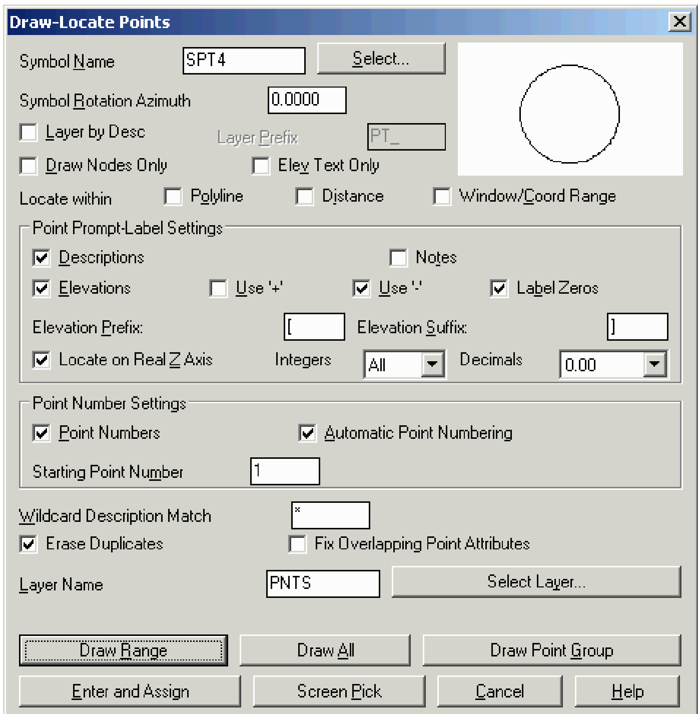
GIS File: This option lets you specify a GIS file to be used when creating new points. The GIS file contains a list of fields to prompt for. For each point that is created, the program will prompt for these fields and store the results

to the note file (.not) associated with the current CRD file.

Pulldown Menu Location: Points
Keyboard Command: ptsetup
Prerequisite: None
File Names: \lsp\ptsetup.lsp, \lsp\scadenvr.dcl

Draw-Locate Points

The Draw-Locate Points dialog box allows you to insert either new or existing points into the drawing. You can create new points either by picking points on the screen, or by entering northing and easting coordinates. You can also place existing points by entering point numbers which reference the current coordinate file. You are prompted to choose a coordinate file if no coordinate file is current.



The name of the symbol file is displayed in **Symbol Name.** You can choose a different symbol by clicking Select. The selected point symbol is displayed on the right.

Symbol Rotation Azimuth is the rotation angle that is used for the point symbols. This angle is used in a counter-clockwise direction relative to the current twist screen.

Layer by Desc inserts the points in the layer named by the point description. Using Layer by Desc organizes the points by description and allows for layer management. For example, you can use the Isolate Layers command to show only points on a certain layer. If you include an invalid layer character in the description, the layer name stops at the bad character. A point description of "UP / 105" would use layer "UP", for example. The Layer Prefix is added to the beginning of the layer name. For example, a Layer Prefix of "PT_" and a point with the description "EP" would use the layer "PT_EP". Layer Prefix is optional. It allows all the point layers to be grouped.

Draw Nodes Only inserts only a point entity (the node) and not the point block and symbol. This option is most useful when you have a lot of points to insert, because inserting only the nodes is faster than inserting nodes with the point block and symbol. Commands such as Triangulate & Contour and Make 3D Grid File can use these points, and do not need the point block and symbol.

Selecting **Elev Text Only** draws text of the point elevation without the point block, symbol, or node. The decimal place of elevation text is placed at the northing and easting point location.

Locate within Polyline inserts only the points that are inside a closed polyline. The command prompts you to select a closed polyline. All the points in the current coordinate file are checked. Any points that are located within the closed polyline are drawn.

Locate within Distance inserts only the points that are within a specified distance from a reference point. The command asks you for the reference point and the search distance. All the points in the current coordinate file are checked. Any points that are located within the search distance of the reference point are drawn.

Locate within Window/Coord Range inserts only the points that are within the specified window or range of northing, easting, and elevation. The command prompts for the minimum and maximum northing, easting, and elevations. These values default to the actual minimum and maximum in the coordinate file. Then the command prompts for the point number range of points to check. The points that fall in both the point number range and the coordinate range are drawn.

Under **Point Prompt-Label Settings**, you determine attributes for which you will be prompted.

Descriptions determines whether you are prompted for descriptions for each point when creating new points. When you are placing both new and existing points, Descriptions determine whether this attribute is labeled with the point inserts.

Notes works with the note file (.not) associated with the current coordinate file. The note file contains unlimited point descriptions in addition to the fixed 32-character point descriptions in the coordinate file. When you create points with Notes on, the command will prompt for point notes to be stored with the point. When you draw existing points with Notes on, any notes for the points are drawn as text entities below the point description.

Elevations determines whether you are prompted for elevations for each point when creating new points. When you are placing both new and existing points, Elevations determine whether this attribute is labeled with the point inserts.

Use '+' labels the positive elevations with a leading '+'. For example, "+159.43".

Use '-' labels the negative elevations with a leading '-'.

Locate on Real Z Axis determines if the points are placed at their elevations or at zero elevation.

Label Zeros will label points with zero elevation when the Elevations option is on. Otherwise only points with nonzero elevation will be labeled.

Elevation Prefix/Suffix set the prefix and suffix labels to apply for the elevation labels.

Elevation Integers controls the number of digits to display to the left of the decimal point for the elevation label. The All setting will show the full elevation digits. The other settings allow you to limit the number of digits to display for the purpose of reducing the amount of space the elevation labels take up in the drawing. For example, if a site is in the 4000 foot elevation range, then this setting could be set to three digits (000) and an elevation of 4321 would be labeled as 321.

Elevation Decimals sets the number of decimals to the right of the decimal places for the elevation labels.

Under **Point Number Settings**, you determine how points will be numbered.

Point Numbers determines whether the complete point block is drawn or just the symbol and node. When you create new points with Point Numbers off, no points are stored in the current coordinate file, and only the point symbol and node are drawn. When you draw existing points with Point Numbers off, the point attribute block is not drawn and only the point symbol and node are drawn.

Automatic Point Numbering applies to creating new points. With this option active, the command will use the **Starting Point Number** for the first new point. The next point number is automatically incremented. Before storing the point, the command checks whether the point number is used. If the point number is used and point protect is on (set in the Coordinate File Utilities command), then the command will prompt for another point number or to overwrite the point. With Automatic Point Numbering off, the command will prompt for the point numbers.

Determine how the points are to be displayed and in what layer.

With **Wildcard match of pt description**, you can display only points with specific descriptions. This can be thought of as a filter. For example, entering IP would display only points that are labeled with the description IP, or Iron Pin. The default is the asterisk (∗). This will display all points regardless of description.

Layer Name allows you to designate a layer for the points to be displayed. You can enter a new name or choose an existing layer by clicking **Select Layer**. A Carlson Survey point consists of a block insert with attributes, a point symbol, and a point entity. The point entity is used for picking the point by OSNAP Node in other commands. The block insert includes a point number, elevation, and description. These attributes are in the PNTMARK, PNTNO, PNTELEV, and PNTDESC layers. The points are also in an overall layer as specified in this dialog box. This layer setup allows you to freeze a group of points by the main layer name or freeze point attributes for all the points in the drawing. For example, freezing layer "PNTS" would freeze all the points in this layer. Freezing layer "PNTELEV" would freeze the point elevation attribute for all the points.

The **Erase Duplicates** option will erase existing point entities that match the point numbers currently being drawn.

Fix Overlapping Point Attributes will detect point number, elevation and description attributes that overlap with other points. Rules can be applied to rearrange the point attributes to avoid the overlaps. A point overlap manager then steps through each overlap for review or manually moving the attributes.

Draw Range will draw existing points from the current coordinate file. The Draw Range button will prompt for the point numbers to draw.

Draw All will draw all the points in the coordinate file, and then zoom the extents of the display to show the points.

Draw Point Group will draw a point group with settings that are established in the Point Group Manager.

Enter and Assign can be used to create new points using the point northing and easting.

Screen Pick allows you to create points by picking the point coordinate on the screen. For example, you could set the Object Snap to EndPoint and pick the end point of a building polyline to create a point at the building corner.

Prompts

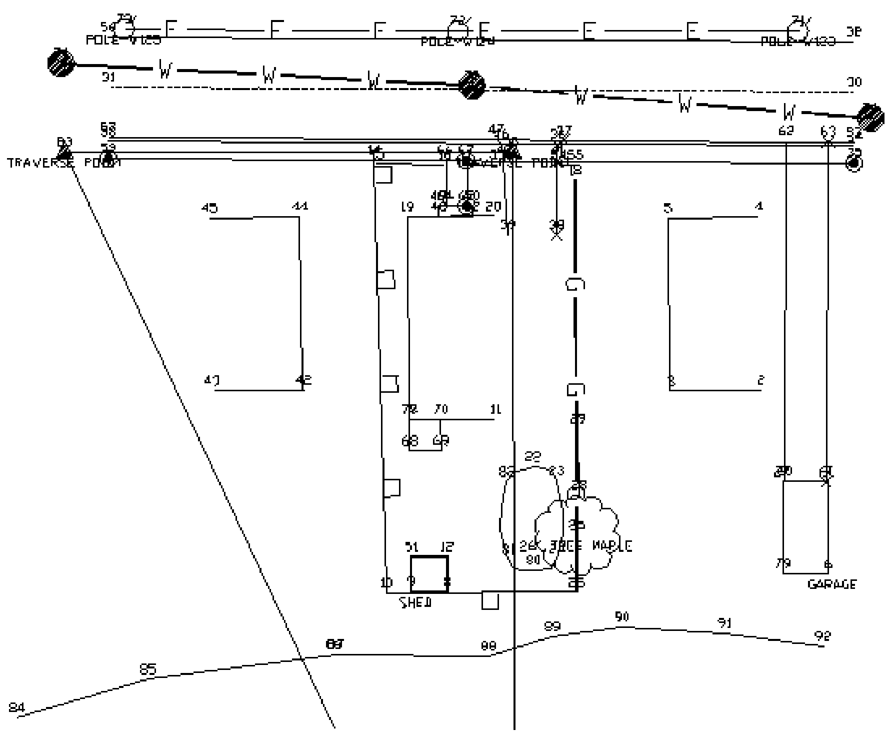
To create a new point:
Draw-Locate dialog *choose Screen Pick*
Pick point to create: *pick a point*
Select/<Enter Point Elevation <0.00>: *Enter elevation* Press S to select text to set elevation.
Enter Point Description <>: *Enter*
N: 5106.57 E: 4901.96 Z: 0.00
Enter/<Select text of elevation>: Select text entity that defines elevation of point.

To locate a point in the coordinate file (point number 3 in this example):
Draw-Locate Point dialog *choose Draw Range*
Point numbers to draw: *3*
Points Drawn> 1
Locates point 3.
Point numbers to draw: *1-2*
Points Drawn> 2
Locates a range of points. From 1 to 2.
Point numbers to draw: *Enter*

Keyboard Commands: lpoint
Prerequisite: A CRD file and you may want to execute *Drawing Setup* (see the Setting menu) to set the scale and size.
File Names: \lsp\lp.lsp, \lsp\crdutil.arx

Field to Finish

This command turns data collector field notes into a final AutoCAD drawing by matching the descriptions of the field points with user-defined codes. The points are brought into the drawing with attributes defined by the code, including the layer, symbol, size and linetype. Draw Field to Finish also uses an improved coding method.



Example drawing results using the example points and example code definitions

Two files are used in Draw Field to Finish - a coordinate file and a field code definition file. The coordinate file consists of x,y,z points with text description fields. The description fields contain codes for the Draw Field to Finish processing. The coordinate file can be a Carlson coordinate (.CRD) file, C&G CRD file, C&G CGC file, Land Desktop MDB file or Simplicity Systems ZAK file. An ASCII data file can be converted into a coordinate file using the *Import Text/ASCII File* command. The field code definition file defines the layer, symbol, size and other actions

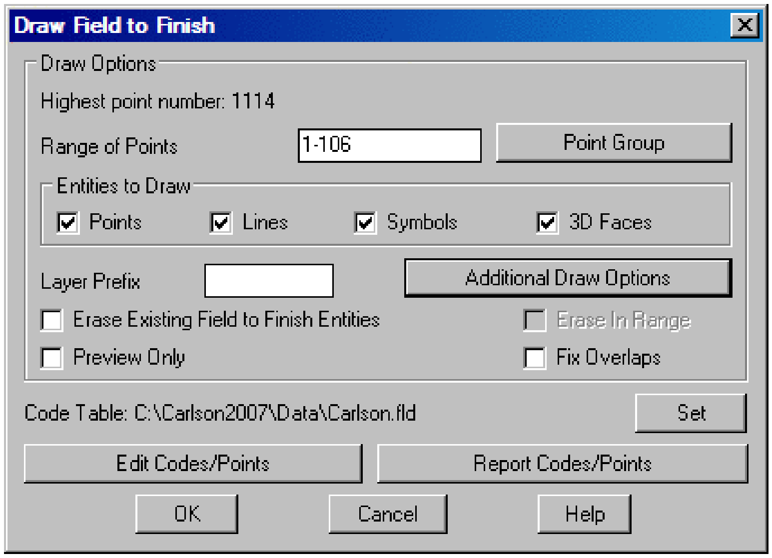
to apply with each code. These file names are displayed at the top line of the Draw Field to Finish dialog box.

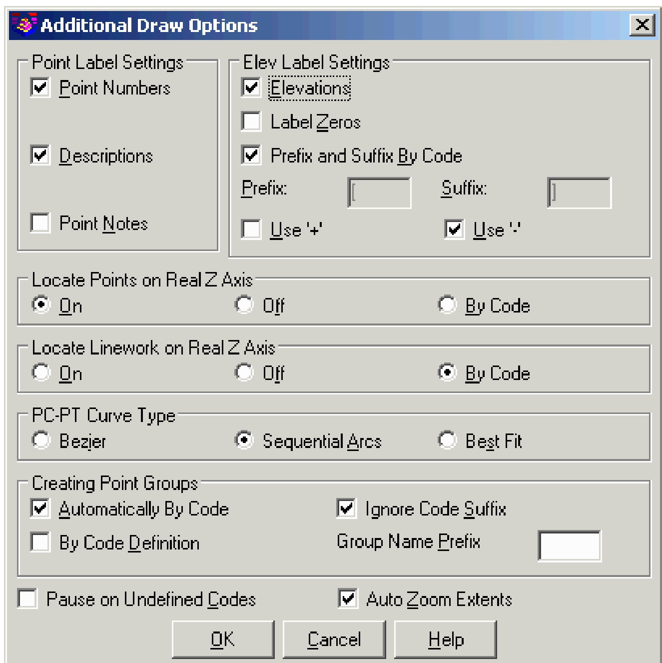
Draw Field to Finish can translate the field points into Carlson points (also called coordinate geometry points or cogo points) with a symbol, layer, and size defined by the code. The point settings of whether to label the description, point number, and elevation and whether to locate the point at zero or at the real Z can be found in the Additional Draw Options of the Draw Field to Finish dialog box. The *Draw-Locate Points* command has these point settings stored separately in the Point Defaults menu. *Draw-Locate Points* provides a simpler method for drawing points compared with *Draw Field to Finish*.

There are two different methods for connecting linework. One method creates line work by connecting points with the same code. The linetype is defined by the code as either points only (no line work), lines, 2D polylines, both 2D and 3D polylines, or 3D polylines (breaklines). Distinct lines with the same code are defined by adding a group number to the end of the code name in the data file. With this method, all points with the description CODE1 will be one line while points with CODE2 will be another line. Both CODE1 and CODE2 use the definition for CODE. For example, the code EP could be a code for edge of pavement that is to be connected as 3D polylines. If there are two separate edge of pavement lines on the left and right sides of a road, all the points for the left side could have the description EP1 and the points on the right side could be EP2.

The second method is the PointCAD format. This method also connects points with the same code. The difference is that instead of using a number after the code for distinct lines, you use the same code with an additional code for starting and ending the line. For example, +0 is used to start a line and -0 to end. So the coding for a segment of edge of pavement could be EP+0, EP, EP, EP-0. Another special code that has been added to Field to Finish is +7, -7. This 7 code will use the linetype definition of line, 2D polyline or 3D polyline defined by the Draw Field to Finish code. For example, if EP is defined as a 3D polyline, then the coding EP+7, EP, EP, EP-7 will create a 3D polyline. Otherwise codes like +0, -0, which is defined as start and end line, will draw EP as a line. Other PointCAD special codes are: +4 starts a curved 2D polyline, ∗4 starts a closed curved 2D polyline, +1 begins a 3-point arc, +5 starts a 3D polyline, ∗5 starts a closed 3D polyline, +6 starts a 2D polyline, ∗6 starts a closed 2D polyline, +7 starts a line whose type is specified by the field code definition, -05 starts a curved 3D polyline section, -50 ends that section, +8 starts a 2D and 3D polyline combination, ∗8 starts a closed 2D and 3D polyline combination, -08 starts a 2D and 3D polyline combination curved section, -80 ends that section. //, followed by a field code, concatenates that field code's description on to the point's description. For example, OAK//04 might become LIVE OAK TREE 4" if the field code OAK translates to LIVE OAK TREE and the field code 04 translates to 4".

The advantage to the PointCAD method is that you don't have to keep track of line numbers. For example, if you are surveying 50 curb lines, the first method would require you to use 50 distinct curb numbers. The advantage to the first method is that you don't have to use the start and end codes. Also the Nearest Found connection option applies to the first method.





Range of Points: Specify the range of points to draw.

Point Group: Specify the point group(s) to process.

Entities To Draw: The Points option draws only the points and point attributes. The Lines option draws only the linework and the Symbols draws only the symbols. Any combination of these options can be processed as well as individual processing of each entity.

Point Label Settings: Specify whether you want Draw Field to Finish to label the Point Numbers, Descriptions, and/or Points Notes which are contained in the note (.NOT) file that is associated with the coordinate (.CRD) file.

Elevation Label Settings: Specify the elevation labeling options. The Label Zeros option will label the elevations of points with z=0. Use Parentheses will place parenthesis around the elevation text. Use '+' and Use '-' will place the appropriate symbol in front of the elevation.

Locate Points on Real Z Axis: Choose between locating all the points at real Z elevation, all at zero elevation or to use the real Z setting as defined in the individual codes.

PC-PT Curve Type: Sets the method for drawing curves with more than 3 points. The Bezier option draws a smooth polyline through all the curve points. The Sequential Arcs method draws multiple arcs with arc end points at each of the curve points. These arcs are tangent to the preceding line segment. The Best Fit method creates a single best-fit curve for all the curve points between the PC and PT.

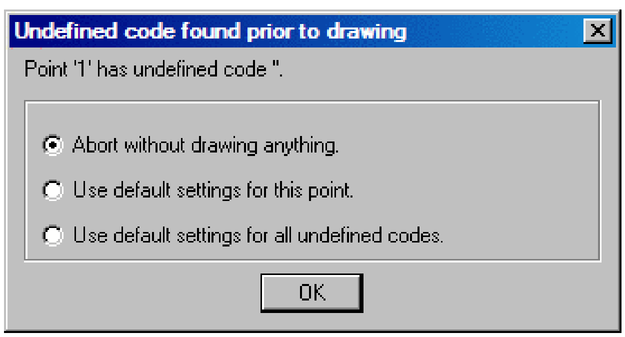
Layer Prefix: Optional layer prefix added to all entities drawn with Draw Field to Finish.

Erase Existing Draw Field to Finish Entities: When checked, this option will erase from the drawing any old entities created by previous Field-To-Finish runs before drawing the new entities.

In Range: This option only erases and redraws those Draw Field to Finish entities that are within the specified range of points to process.

Creating Point Groups: Point Groups can be created in one or two different ways. Each field code definition can specify a Point Group that all point numbers that use that code will be added to. Multiple field codes can use the same Point Group name. Check the By Code Definition checkbox for that option. The second method is to automatically create Point Groups for each code that is processed. Check the Automatically By Code checkbox for that option. Ignore Code Suffix, if checked, will cause the codes to be considered after removing the numeric suffix. For example, points with the EP10 and EP11 codes will both be automatically added to the Point Group named EP. No matter how the Point Group is created, the Group Name Prefix can be used to add a prefix to the group name. Note: if the Point Group already exists, it will be erased first before being created again by either of these two methods.

Pause on Undefined Codes: When checked, Draw Field to Finish will pause if it encounters a description that is not defined in the code table.



Abort without drawing anything: This stops the command. Run Draw Field to Finish again to correct the code table.

Use the default settings for this point: This option draws a point in the "MISC" layer with no linework. To set your own default, define a code called "SC_DFLT".

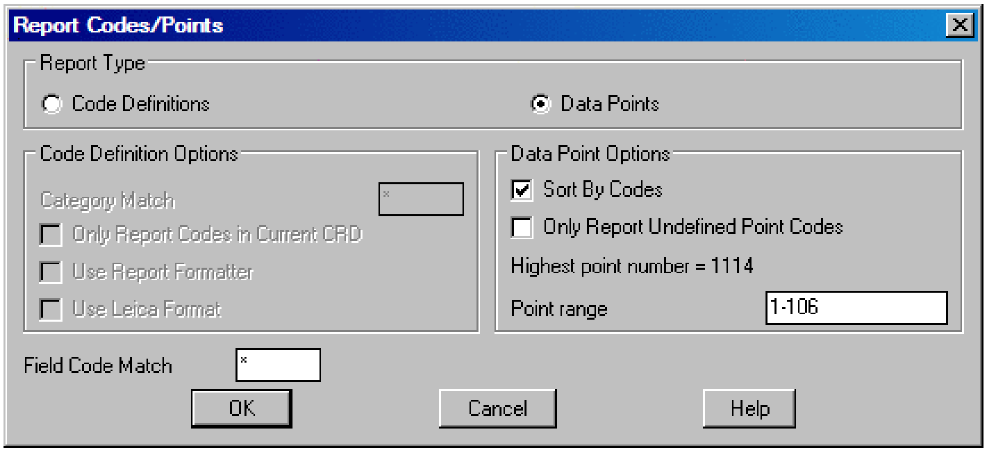
Use default settings for all undefined codes: This option will draw all undefined codes in the "MISC" layer by default or a user specified layer as defined in the "SC_DFLT" code. A good way to check the data file for unmatched descriptions is to use the Print Table command and choose the Data Points and Distinct Code options. This command will print the different codes in the data file and identify any undefined codes.

Draw (continued)

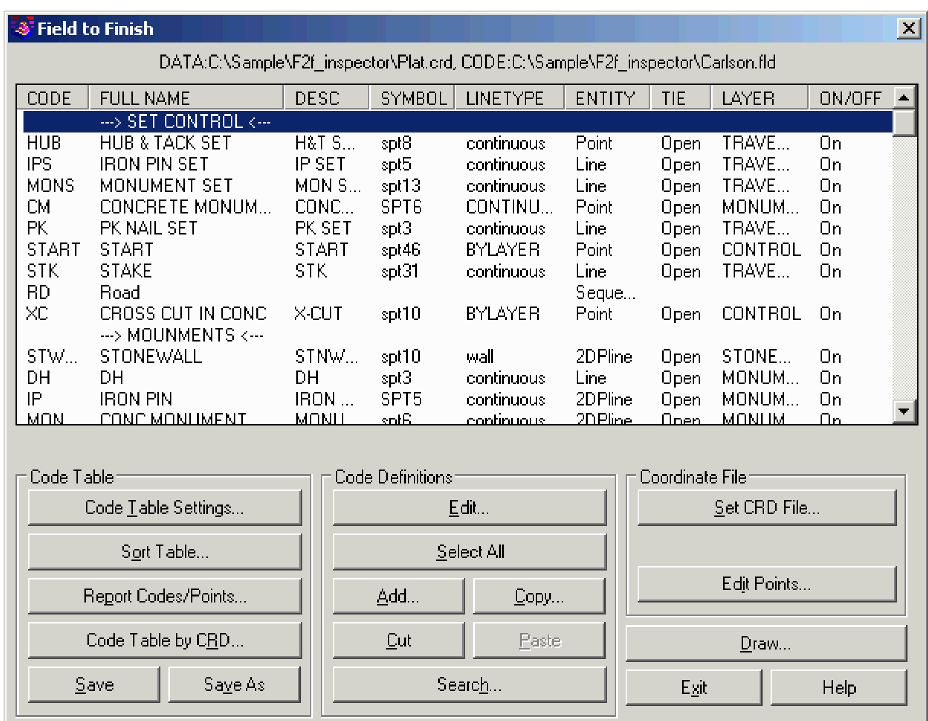
Preview Only: When checked, this option will temporarily draw the points and linework and allow you to review it with zoom and pan.

Auto Zoom Extents: When checked, this will force a zoom extents after Draw Field to Finish is done.

Report Codes/Points: This routine prints the code table or the data file to the screen, file, or printer. A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.

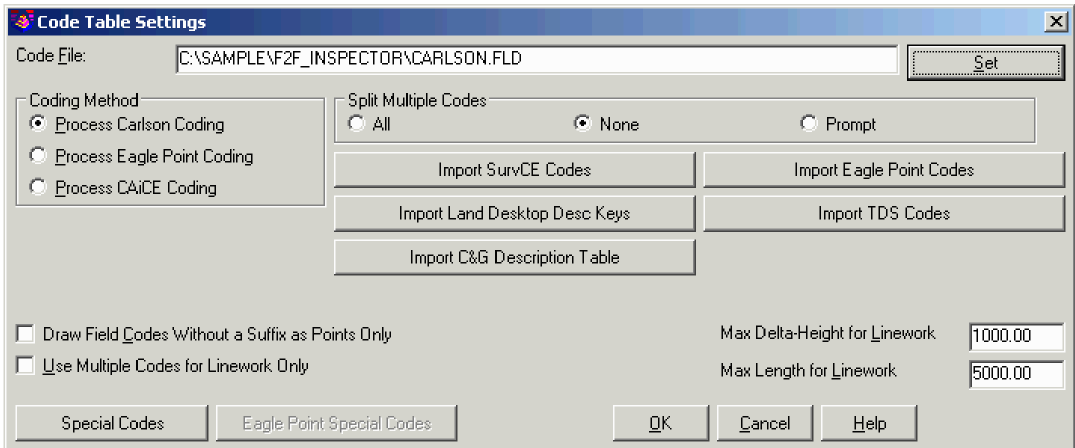


Edit Codes / Points: The Field to Finish dialog box allows you to load the coordinate and field code definition files, view and edit the code definitions, view and edit the coordinate file, view reports, and then return to the Draw Field to Finish dialog box to process the files. The top section displays the code definitions. The bottom section has three columns of functions each pertaining to controls for different elements of the command. The **Code Table** section provides controls for settings, sorting and reporting of codes. The **Code Definitions** section provides tools for the creation and editing of codes. The **Coordinate File** section provides controls for coordinate files and points. It also contains the Draw controls which starts the processing of the data using Draw Field to Finish.



Code Table

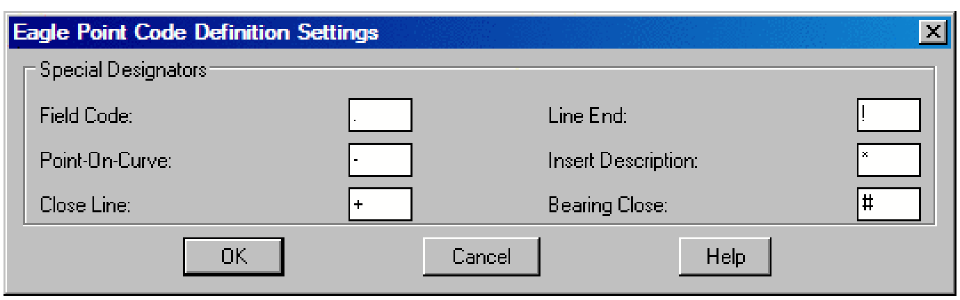
Code Table Settings: These options provide tools for defining the coding method to be used for processing of the point data. Various import tools allow for the importing of codes from different software packages. Controls for handling multiple codes are located on this dialog. All special codes can be replaced to other characters defined by the user. The special codes are listed and edited on this dialog.



Set: Choose this button to specify a new code table. The name of the current table is shown in the field to the right of this button.

Process Carlson Coding: When checked, this option interprets and processes coordinate files based upon the Carlson Coding method and data collection method.

Process Eagle Point Coding: When checked, coordinate files are processed based on the Eagle Point Data Collection method. When selected the ***Eagle Point Codes*** button becomes available for selection and displays the following dialog. This dialog allows for customization of the eagle point special designators.



Currently the supported designators include, "Field Code", "Point-On-Curve", "Close Line", "Line End", "Insert Description" and "Bearing Close". Also supported is the ability to recognize overwriting of descriptions just as Eagle Point does by using the space separator instead of the "Insert Description" designator. Examples of supported coding are as follows:

.TC Places a node and or line per the field code library.
TC Places a node and or line per the field code library.
-TC Specifies a point on a curve.
TC- Specifies a point on a curve.
..TC Stops the line.
TC! Stops the line.
.TC+ Closes the line back to the starting point.
TC+ Closes the line back to the starting point.
.TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.

TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.
WV.W1 Places a node as specified by the code "WV" in the field code library and then begins a line as specified by code "W" in the field code library.
.TC.EP.FL Results in three lines coming together.
TC1.TC2.TC3 Results in three lines coming together. All three lines are specified by the definition of the single code "TC" in the field code library.
TC.TC1 When used in conjunction with the "Draw Field Codes Without a Suffix as Points Only" toggle, "TC" will be recognized as the node and "TC1" will be recognized as the line so that if the code "TC" in the field code library is defined as a polyline, line or 3D polyline, duplicate lines will not be unintentionally placed when this shot only pertains to a single element. Keep in mind that all line work must have a numeric suffix when using this toggle.
TREE ∗ OAK Result on screen would be: TREE OAK
TREE OAK ∗ Result on screen would be: OAK TREE
TREE OAK Result on screen would be: OAK
TC1!.TC2-.VLT6# Stops "TC1", continues "TC2" as a point on a curve and closes VLT6 as a rectangle using the "Bearing Close" code.

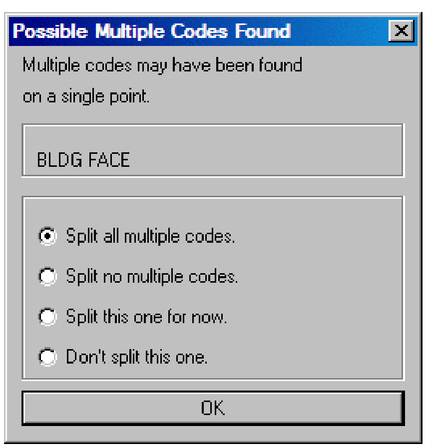
Note: The use of the "Use Multiple Codes for Linework Only" toggle is recommended when using Eagle Point Coding.

Process CAiCE Coding: When checked, coordinate files are processed based on the CAiCE Data Collection method. Examples of supported coding are as follows:
169 is just the code 169.
145C10 is the code 145 and line #10.
169C25C is the code 169, line #25, and the point is on a curve.
172C12B is the code 172, line #12, and this point closes the line.

Split Multiple Codes:

Multiple codes are defined by including each code in the point description field separated by a space. A single data point can be used in different lines by assigning it multiple codes. For instance, a point might be part of both a curb line and a driveway line with a description of "CURB DRW". Field-to-Finish uses spaces as the delimiter for multiple codes. You should avoid spaces in the descriptions except for where multiple codes are intended or after the "/" character. For example, a code for light post should not be "LGT POST" but instead should be "LGTPOST".

There are three options for the handling of multiple codes when encountered. The **All** option will split all multiple codes and process each code based upon their code definition. When **None** is select both codes will be processed based upon their code definition. If the **Prompt** option is checked on, when Field-to-Finish detects multiple codes on a point the following dialog will be displayed with options for handling the codes.



Import Land Desktop Desc Key: This option imports and converts a Land Desktop Description Key into a Carlson Draw Field to Finish (fld) code definition file. The Land Desktop Description Key file is a mdb file and is found in the Land Desktop Project file path. It is located in the under the COGO/DescKey directory.

Import TDS Codes: This option imports TDS codes into the Carlson Field to Finish (fld) code definition file.

Import Eagle Point Codes: This option imports Eagle Point codes into the Carlson Field to Finish (fld) code definition file.

Import C&G Description Table: This option imports C&G code tables (tbl) into the Carlson Field to Finish (fld) code definition file.

Import SurvCE Codes: This option imports a SurvCE Feature Code List (fcl) into a Carlson Field to Finish (fld) code definition file.

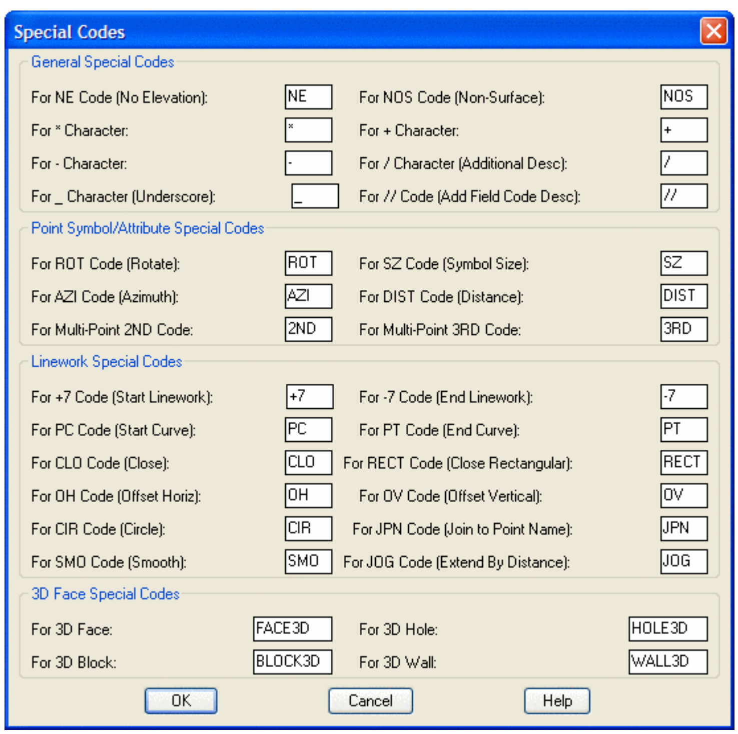
Draw Field Codes Without a Suffix as Points Only: This option is useful for when wanting to use a field code sometimes for linework and sometimes for just points but it is preferred to number the lines rather than using start and stop codes. For example, if the field code EP is defined to use the Line Entity type, then EP25 will be drawn as a Line, however if just EP is used, no linework will connect to that COGO point.

Use Multiple Codes for Linework Only: When checked, and multiple codes are detected, only linework will be drawn for the secondary codes. Points are only created based on the primary code. If you want symbols for all multiple codes, then this setting should not be checked.

Max Delta-Height for Linework: Use this option to specify the maximum elevation difference that Draw Field to Finish should draw any section of linework. This option is for use with 3d polylines and lines.

Max Length for Linework: Specify the maximum length that Draw Field to Finish should draw any section of linework.

Special Codes: This section allows you to substitute the existing predefined special codes and characters with your own. Draw Field to Finish recognizes several special codes. A special code is placed before or after the regular code with a space separating the code and special code. Here is a listing of the default special codes and characters.



Special Characters

The characters (∗, -, +, /, and _) can be used and substituted in Draw Field to Finish. The way these characters are used is that when the file is processed the description field is searched for these characters. If the "+" symbol was changed to "-" then the program would look for "-" and change it to "+". This is useful when a particular data collector may not have all the symbols available. With these substitutions you can make a character that is provided on the data collector generate the symbol needed. Multiple characters can also be used. For example "–" can be used to in order to produce a "/" character or any of the characters listed above.

Special Codes

"/"

Carlson points in the drawing have point attributes including a description. When Field-to-Finish draws the points, the point description from the coordinate file is processed to match a code. The code then defines the description that is drawn with the point. For example, consider a code of "UP" with a description of "POLE" and a data point with the description "UP". The data point description "UP" would be matched with the code "UP" and the point would end up being drawn with the description "POLE". A special character "/" (the forward slash or divide key) can be used for an unprocessed description to append. Everything after the "/" is added directly to the point description and is not considered a code and no further substitution is done on it. For example, a data point with the description "UP / 150" with the same code "UP" definition above would be drawn with the description "POLE 150".

"//"

This special code causes text after the "//" to be interpreted as a field code. That field code's description is then appended to the first field code's description. For example, if the field code 02 has the description *2"* and the field code OAK has the description *oak tree*, then 02//OAK will result in the point having the description of *2" oak tree*. If the "/" character has been replaced with a different character, for example with a & character, then the "//" code would become "&&".

PC

This code begins a three point arc or a curved line when used with the "PT" code (see below). The point with this special code is the first point on the arc. The next point with the code is considered a point on the arc, and third point with the code is the arc endpoint. For example (in point number, X, Y, Z, description format),

10, 500, 500, 0, EP PC - start curve
11, 525, 527, 0, EP - second point on curve
12, 531, 533, 0, EP - end point of curve

PT

This is a special code that can be used with "PC" to define a curve with more than three points or a tangent two-point curve. Starting at the point with the "PC", the program will look for a "PT". If the "PT" is found, all the points between the "PC" and "PT" are used for the curve which is drawn as a smoothed polyline that passes through all points and only curves the polyline between points. If no "PT" is found, then the regular three point arc is applied as explained above. If no points are found between the "PC" and "PT", then the point prior to the "PC" and the point after the "PT" are used to create tangents for the resulting curve.

CLO

This code forces the lines drawn between a series of points with the same code to close back to the first point with the same code. For example, shots 1-4 all have the BLD description with the exception of point 4. Its description is BLD CLO. This will force the linework drawn for the BLD code to close back to point 1 which is the first point with the description of BLD.

NE

This code represents no elevation. A point with this special code is located at zero elevation.

NOS

This code indicates that the point should be "non-surface"; that is, that it should be ignored when contouring or creating surfaces. This can also be controlled per-field code by turning on the Non-Surface toggle in the Edit Field Code Definition dialog box.

OH & **OV**

The codes "OH" and "OV" stand for offset horizontal and offset vertical. These offset codes apply to 2D and 3D polylines. A single set of offset codes can be used to offset the polyline a set amount. For example,

10, 500, 500, 100, EP OH2.5 OV-.5
11, 525, 527, 101, EP
12, 531, 533, 103, EP

This would create a polyline connecting points 10,11 and 12 and an offset polyline with a 2.5 horizontal and -0.5 vertical offset. The direction of the horizontal offset is determined by the direction of the polyline. A positive horizontal offset goes right from the polyline direction and a negative goes left. The horizontal and vertical offset amounts apply starting at the point with the offset codes until a new offset code or the end of the polyline. Only one horizontal and vertical offset can be applied to 2D polylines. For 3D polylines, multiple offset codes can be used to make a variable offset. For example,

10, 500, 500, 100, EP OH2.5 OV-.5
11, 525, 527, 101, EP OH5.5 OV-.75
12, 531, 533, 103, EP OH7.5

This would offset the first point horizontal 2.5 and vertical -0.5, the second point horizontal 5.5 and vertical -0.75 and the third point horizontal 7.5 and vertical -0.75.

SZ

This code is used to set a different symbol size. The value of the new symbol size is specified after the SZ (example SZ0.2). This value is a size scaler that is multiplied by the current drawing scale to determine the actual drawn size. For example, a drawing scale of 50 and a symbol size scaler of 0.2 would make the drawn symbol size 10. Two dimensional scales can be accomplished by using an ‘X' between the horizontal and vertical scales (e.g., "SZ0.2X3.5"). If no number follows the SZ special code, then the next point with the same field code as the current point will be used to determine the scale factor.

ROT

This code is used to set the rotation of the point symbol. If a point number follows the ROT code, then angle from the current point to this point number is used for the rotation. For example, "ROT45" would rotate the symbol towards point number 45. If there is no point number after the ROT code, then the rotation point is the next point number with the same code as the current point. ROT can also be used to rotate towards an angle clockwise from north by using ‘+' or ‘-' in front of the number. For example ROT+45 rotates the point symbol to the northeast and ROT-90 rotates the point symbol to the west.

SMO

This code is used to smooth the polyline.

AZI & DIST

The AZI and DIST codes are used together to locate an offset point. The AZI sets the offset azimuth and DIST sets the distance. The values should directly follow the code. For example, AZI25 DIST4.2 would draw the point offset 4.2 at an azimuth of 25 degrees.

JOG

The "JOG" special code allows for additional points to be inserted into the line work at perpendicular or straight offsets. Only offsets should follow the JOG code. Positive numbers indicate a jog to the right and negative numbers indicate a jog to the left. Alternatively, "R#" and "L#" can be used where # is the distance to either the right or the left. Finally, "S#" can be used to make an offset straight ahead by using a positive # or behind by using a negative #. For example, "BLDG JOG S10.1 R5 L12.2 L5 L12.2" or equivalently "BLDG JOG S10.1 5 -12.2 -5 -12.2" advances 10.1 units and then draws a closed rectangle on the right hand side of an existing line. The offsets are always done in the X-Y plane. If the current line is vertical, an offset to the right is along the positive X-axis.

JPN

The "JPN" (Join to Point Name) special code joins to the point named immediately after the code. For example, "JPN205" causes a line to be drawn from the current point to the point "205".

RECT

The "RECT" special code causes a rectangle to be formed on a 2D or 3D polyline using one of two different methods. If a number follows "RECT" (e.g., "RECT10"), a rectangle will be drawn 10 units to the right of the last two points ending on the point with the "RECT" code. Use a negative offset to place the rectangle on the left side (e.g., "RECT-2.5"). For example if locating the left side of a 10' rectangular concrete pad using the code conc for concrete, the description of the two left points would be (conc) for the first point and (conc rect10) for the second. If no number follows "RECT", then the polyline will be closed by shooting right angles from the first point of the polyline and the current point and creating a new point where those two lines cross. This method requires three points be established on the pad.

CIR

The "CIR" special code stops the linework on the previous point and causes this point to create a circle in one of three different ways. The first way uses just the current point as the center with the CIR special code followed immediately by the radius. For example "CIR5.0" will create a circle centered on this point with radius 5 and at the elevation of the current point. The second method uses two points, the first point specifying the center and the elevation, and the second point specifying the radius. The third method uses 3 points that specify the perimeter of the circle in 2D with the first point specifying the elevation. The "CIR" code can be used with all of the linetypes including "points only". The circles are always parallel to the X-Y plane.

For Multi-Point 2ND Code

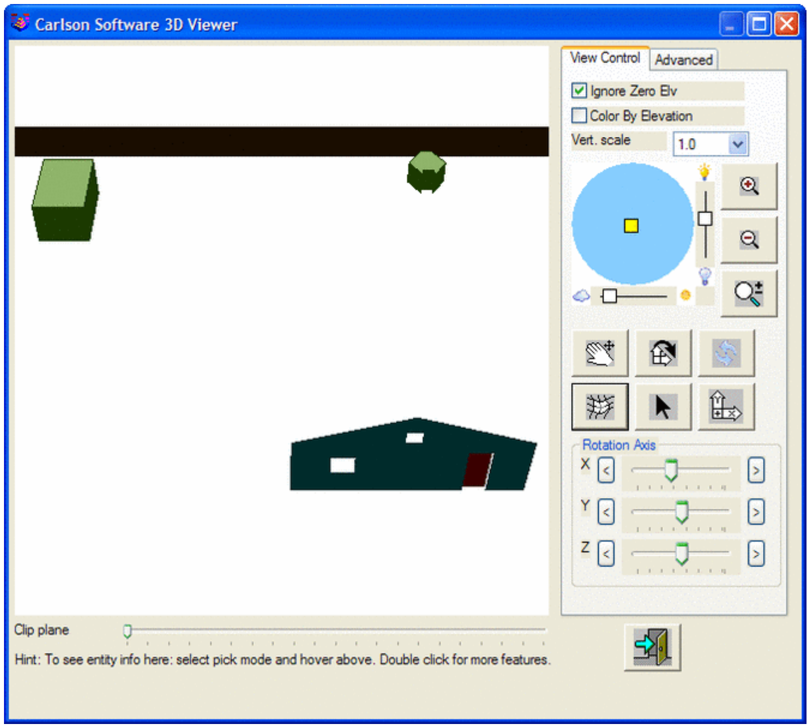
When used on the first point of a multi-point symbol, the "2ND" code indicates that the second point of the sequence (i.e., the next point after the current one) should be used as the second symbol insertion point for a multi-point symbol. Please refer to Symbol Pts in the Edit Field Code Definition section below.

For Multi-Point 3RD Code

When used on the first point of a multi-point symbol, the "3RD" code indicates that the third point of the sequence should be used as the third symbol insertion point. The "3RD" code should be used with the "2ND" code. Please refer to Symbol Pts in the Edit Field Code Definition section below.

3D Special Codes

Below are the special codes that can be used for the easy creation of 3D surfaces. The resulting AutoCAD 3D face entities can be viewed in the Carlson 3D viewer by entering "cube" on the command line.



FACE3D

Makes a triangle mesh of 3D face entities by triangulating points starting with the current point and continuing until the line ends or another 3D special code is found. The points must be ordered along the perimeter. Although the mesh will be built if the points are clockwise or counterclockwise along the perimeter, the visible side in the Carlson 3D viewer, "cube", is the clockwise side by default. On the Advanced tab, the shading mode may be set to *Shade both* or *Shade back* if you would prefer to see both sides or just the counter-clockwise side.

HOLE3D

Makes an exclusion area within the triangle mesh identified by the point number following this code (e.g., "HOLE3D101" will start a hole in point # 101). If no point number is given ("HOLE3D"), the exclusion area is applied to the last mesh or if there is a mesh in the process of being constructed by the current sequence of points, it is ended and the hole is applied to it. Note that a hole can only be applied to a mesh that was created by FACE3D (not BLOCK3D or WALL3D). Note also that it can be difficult to predict what the "last mesh" was if it used a different field code since the points of the coordinate file are processed by order of field code first and then point number. There is no limit to how many holes can be applied to a FACE3D mesh. The points of the hole itself are not added to the FACE3D mesh; they are projected on to the best plane that contains the FACE3D mesh and then the hole is cut-out.

Example 1:
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 VENT1 HOLE3D2500 */applies 2505-2508 as a hole to last mesh that uses point #2500. So any point in the range 2500-2504 would have the same effect.*
2506 VENT1
2507 VENT1

2508 VENT1

Example 2:
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 HOUSE1 HOLE3D */stops the above mesh and applies 2505-2508 as a hole*
2506 HOUSE1
2507 HOUSE1
2508 HOUSE1

Example 3:
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 WINDOW1 FACE3D HOLE3D2503 */applies 2505-2508 as a hole to above mesh 2500-2504 and starts a new mesh using the WINDOW field code.*
2506 WINDOW1
2507 WINDOW1
2508 WINDOW1

Example 4 (same result as Example 3):
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 WINDOW1 FACE3D */starts a new mesh using the WINDOW field code.*
2506 WINDOW1
2507 WINDOW1
2508 WINDOW1 HOLE3D2504 */makes the mesh 2505-2508 also be a hole in the mesh 2500-2504.*

BLOCK3D

Makes a set of 3D faces to make a 3d block using the height value entered after the code (e.g., "BLOCK3D2.3" with height 2.3). Heights can be positive or negative. With 3 points, makes a parallelogram base that is extruded up (or down if height is negative) to form a 6-sided block, including top and bottom. With 4 or more points, makes a closed polygon for the base that is then extruded by the height. The points can be laid out in clockwise or counterclockwise order around the perimeter. The perimeter or base does not have to be a convex polygon.

WALL3D

Makes a set of 3D faces above the polyline using a height value entered after the code (e.g., "WALL3D2.3" with height 2.3). The height can be negative if the points on the top of the wall have been shot. If no parameter exists, then the height is determined by the distance from the current point to the next point. This is a signed distance so the surveyor can shoot either the top of the wall or the bottom of the wall. Both sides of the wall will have triangles and so both sides will always be visible in the Carlson 3D viewer "cube".

Example 5 – 6' high wall shot along the bottom:

2000 1000.000 1060.000 100.000 WALL1 WALL3D6.0 */wall 6'*
2001 1100.000 1060.000 100.000 WALL1
2002 1100.000 1160.000 100.000 WALL1

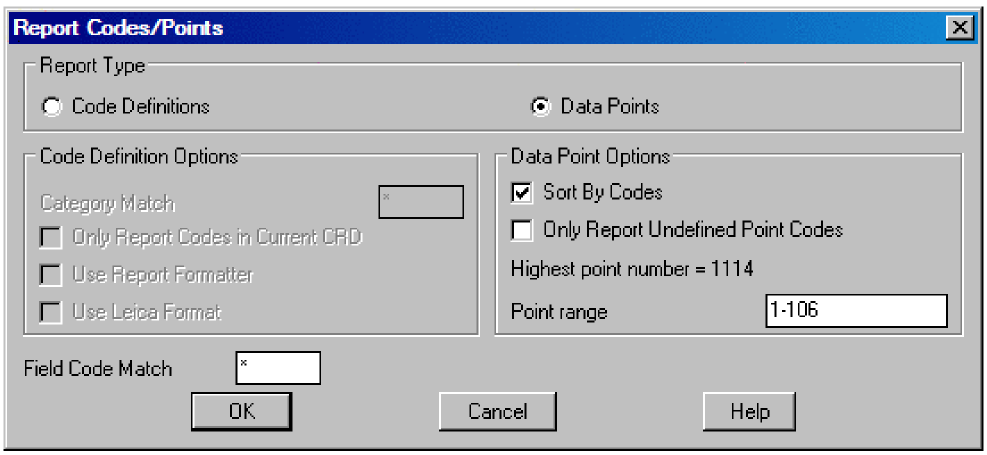
Example 6 – 6' high wall, height specified by 1st to 2nd point, shot along the top:

2020 1100.000 1160.000 100.000 WALL2 WALL3D */height by 2nd pt*
2021 1100.000 1160.000 106.000 WALL2
2022 1000.000 1160.000 106.000 WALL2

Code Table (continued)

Sort Table - This sorts the code table by either code name or layer.

Report Codes/Points - This routine prints the code table or the data file to the screen, file, or printer. A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.



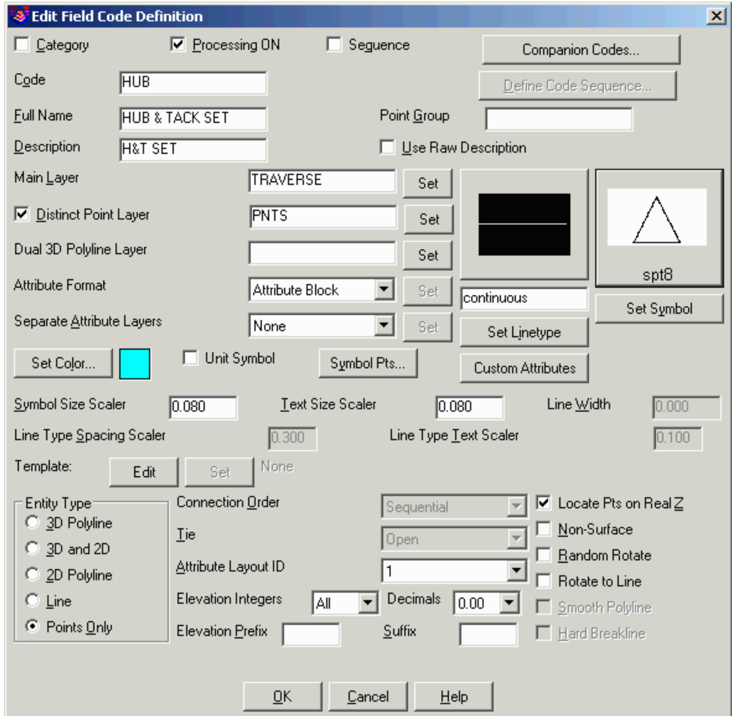
Code Table by CRD - This command will create code table definitions based on the coordinate file field descriptions. This is useful when creating a code table from scratch.

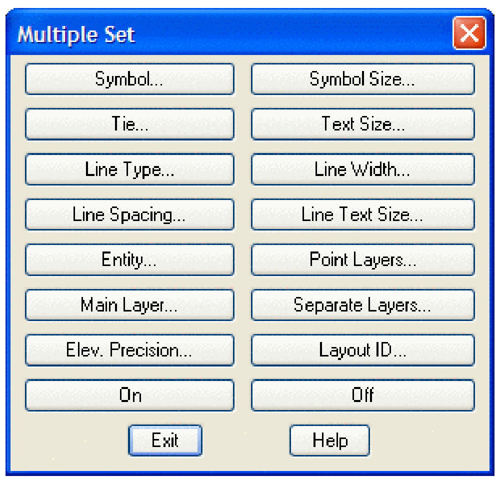
Save: Saves the Draw Field to Finish field code definition (.FLD) file.

Save As: Reacts the same as Save but allows for specification of file name and location to save to.

Code Definitions

Edit: If only one field code is selected, then this command opens the Edit Field Code Definition dialog box. If multiple field codes are selected (by holding down the control key or shift key and clicking on the rows), then the Multiple Set dialog box will open.





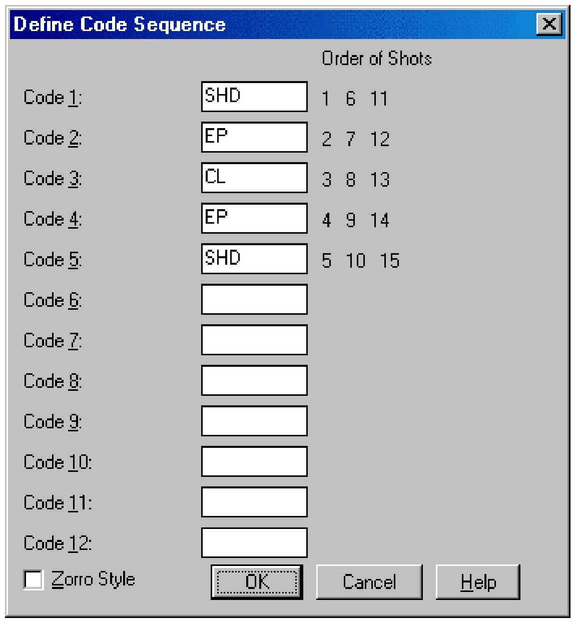
Field-to-Finish will layerize the points and linework according to the code definitions. If the layers to use are not already defined, Field-to-Finish will create the necessary layers and assign different colors. To have the same colors for these layers in all your drawings, define the layers in the prototype drawing. The prototype drawing is the default drawing that is loaded whenever a new drawing is created. To define layers in the prototype drawing, save your current drawing and then start a new drawing with the New command. Don't give the new drawing a name, just click OK. Then define the layers as desired with the *Layer* command. When you are done creating layers, use the Save As command and change to Drawing Template (.DWT) under Save as Type. The default drawing template that is used is named 18SCDRAW.DWT. This template name will correspond to the version of AutoCAD that is being used, for example 16SCDRAW for AutoCAD 2004 users. You can overwrite this default template or make a new drawing template. If you make a new one, you may want to edit the Carlson icon to use the new one. To edit the icon, highlight the icon with one click and then click the right mouse button. Choose Properties and then Shortcut and change the drawing template name.

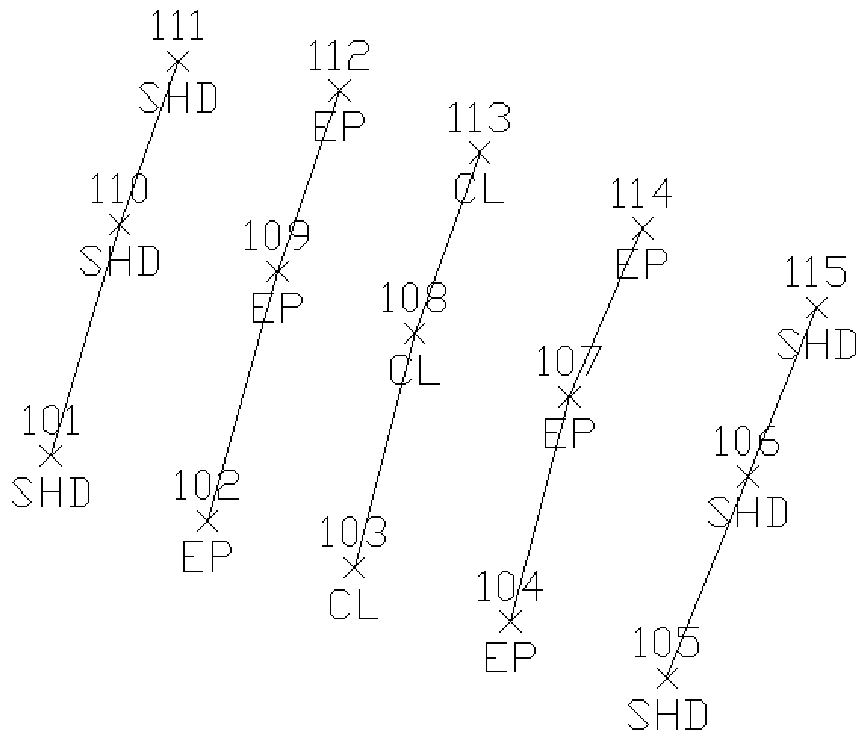
Category: This is an optional field that can to used to help organize your codes. A category is not used for processing and only is useful in viewing and printing.

Processing ON: This toggle controls whether this code will be processed.

Sequence: This specifies a sequence type code. Sequences are a way to simplify field entry of a sequence of codes. For example, a road cross-section could be SHD1 EP1 CL EP2 SHD2. Instead of entering these different descriptions, one sequence definition can store these descriptions in order. Then just the sequence code (such as RD) is used in the field. The cross-section can be shot in left to right then left right order, right to left then right to left order, or alternating left to right then right to left order. The alternating method is known as the Zorro style. The one restriction is that the shots always start from a right or left edge.

To set up a sequence, choose the Sequence toggle in the Edit Code dialog. Then pick the Define Code Sequence button. This brings up a dialog for entering the sequence codes in order. These sequence codes should be defined as normal codes somewhere else in the Draw Field to Finish code table (ie SHD as a 3D polyline). In the field, the one template code is used for all the cross-sections shots (ie RD for all the points). Then Draw Field to Finish will substitute this template code with the sequence codes (ie substitute RD with SHD).

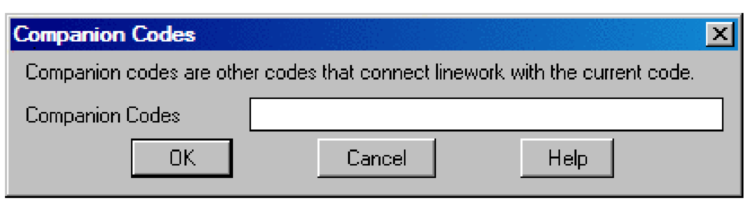




Resulting points and linework showing Zorro style template

Define Code Sequence: This sets the code names that make up the sequence.

Companion Codes: This option allows different codes to connect when defined as line, polyline or 3d polyline. For example, a main line power pole code may be defined as PP while a service utility pole may be defined as UP. When processing Draw Field to Finish, it may be desired to connect all PP and UP codes together. This could be accomplished by defining a companion for UP as PP and a companion code for PP as UP. Each code needs to reference the other as a companion code.



Code: This is the key name that identities the code and is matched with the field data descriptions. It is important to note that the * character, used in this field, is regarded as a wildcard or "match anything" code. For example, a field code definition with the code defined as TREE* will be used for any raw description of TREE. Raw descriptions of TREEA, TREE12, TREE, etc. will match the TREE code definition. This will always be the case unless there is a more specific code is found. For example is there was a code TREEA in the code definition file, then that code would be used instead of the TREE code.

Full Name: This is an optional field that describes the code for viewing.

Layer: The point and line work for the code will be created in this layer.

Description: This value is assigned to the point description attribute when the point is drawn. This description can be different than the field description. An additional description can be added to a point by entering it after a forward slash in the data description field.

Use Raw Description: This option turns off the Description field described above. Instead the points will be drawn with their original unprocessed descriptions.

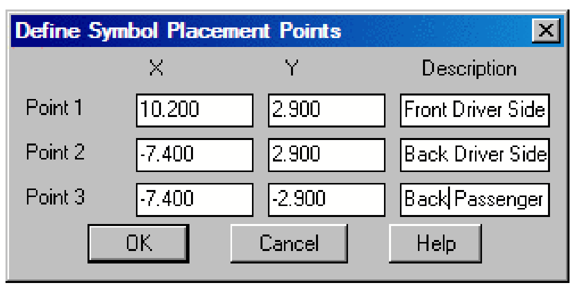
Dual 3D Polyline Layer: Displays the layer that the 3d polyline will drawn on when using an Entity Type of 3D and 2D. The layer name can be typed in this field.

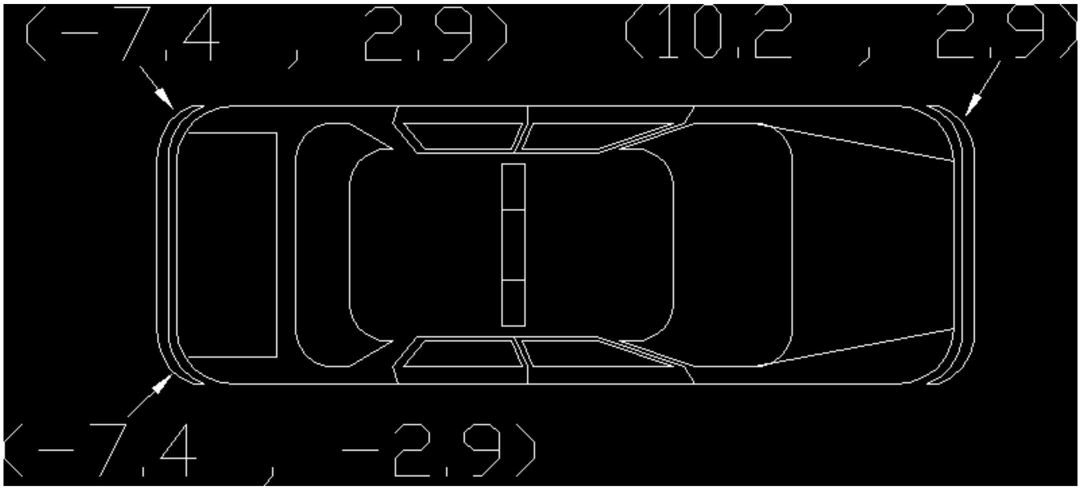
Set 3D Layer: Sets the layer that the 3d polyline will drawn on when using an Entity Type of 3D and 2D. The layer can be selected from the list or typed in at the bottom of the dialog box.

Set Linetype: Line work can be drawn in any of the special linetypes or with the linetype for the layer ("BY-LAYER"). The spacing and size of the special linetypes is determined by the AutoCAD LTSCALE system variable and by the field code settings *Line Type Spacing Scaler* and *Line Type Text Scaler*. The special linetype "hedge" is drawn with a user specified width. The special linetype "userdash" is drawn with user specified distances for the length of the dash and the length of the gap between dashes. You will be prompted for this information when you select that linetype.

Set Symbol: This is the point symbol for the code. To avoid drawing a symbol, use the Carlson symbol named SPT0.

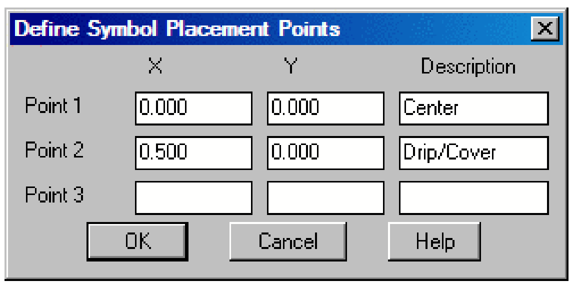
Symbol Pts: For each code definition, the symbol insertion points can be defined with up to three points. To define the symbol insertion points, choose the Symbol Pts button in the Edit Code Definition dialog box. By default, the symbol insertion is defined by one point at the symbol center (0,0). A one point insertion definition can be used to insert a symbol offset from the center. With a two insertion point definitions, the program will rotate and scale the symbol. For example, two insertion points can be used to insert a tree symbol to size the tree, where the first point is for the tree center and the second is for the drip line. With three insertion point definitions, the program will rotate and scale the symbol in both X and Y. For example, three points can be used to insert a car symbol with the first point being the front drivers side, the second point as the back driver side (to rotate and scale the length) and the third as the back passenger side (to scale the width). Besides the insertion point coordinates, you can define a description for each point which is used for the drawn point description and is used for prompting in the Insert Multi-Point Symbol command and in Carlson Field data collection.

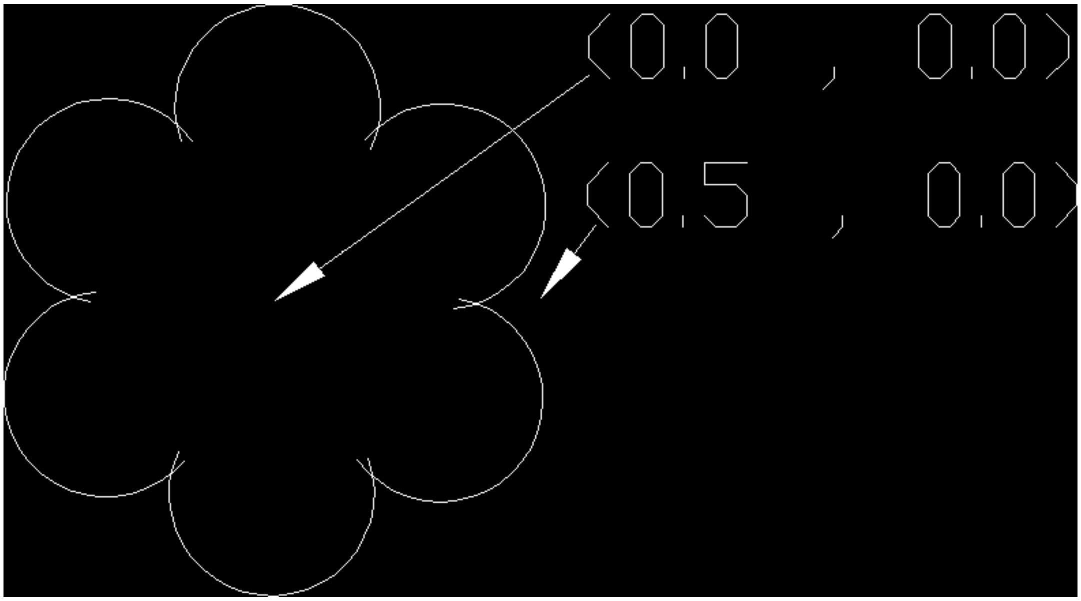




Three Point Symbol Drawing

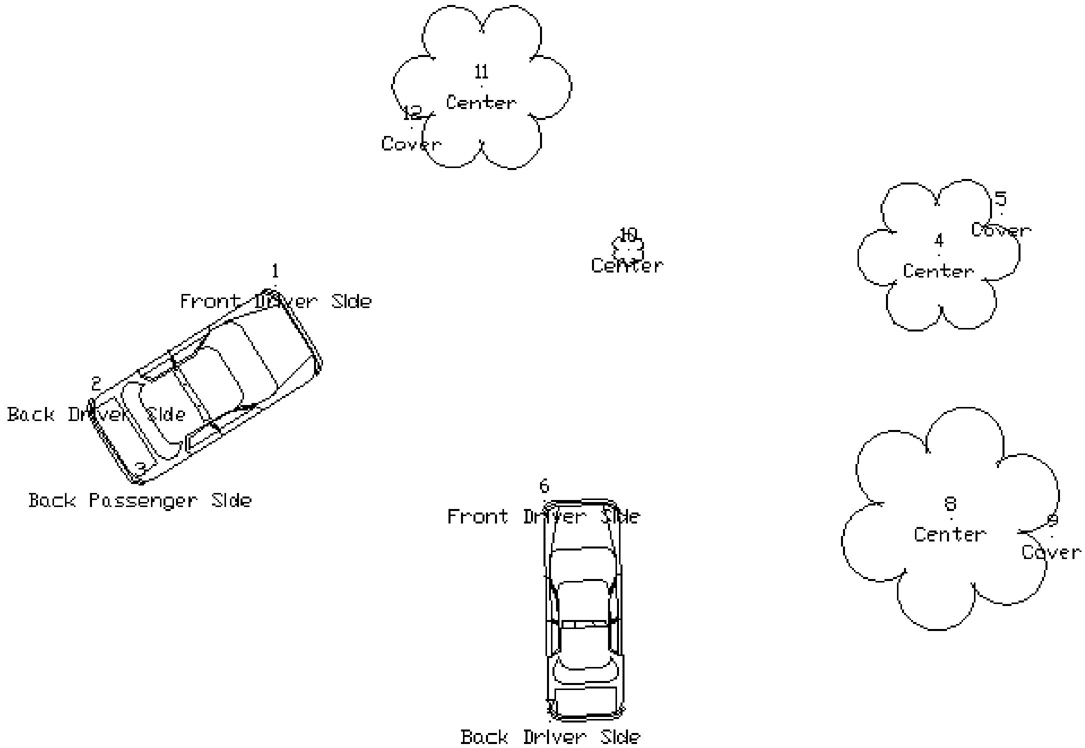
The coordinates for the insertion point definitions are for the symbol at unit size. To figure these coordinates, you will need to open the symbol drawing (.DWG) file. By default, the symbols are located in the Carlson SUP directory. For example to make an insertion point for the tree drip line, open the tree symbol drawing and find the coordinate at the edge of the tree symbol (in this case 0.5,0.0).





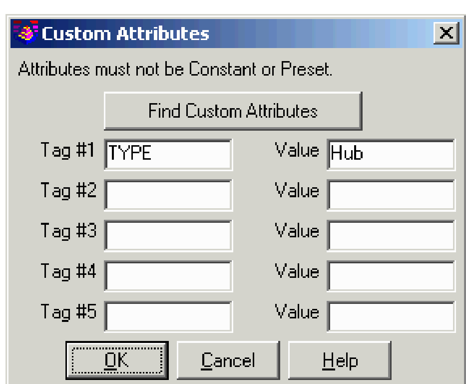
Two Point Symbol Drawing

Not all of the symbol insertion points need to be used when drawing the points. If a code definition has a three insertion points, it is possible to use just the first two or first one. There are special codes to associate multiple points to the same symbol. The first code point is used as the first symbol insertion point. The "2ND" code is used to specify the second symbol insertion point. A point number can follow the "2ND" to identify a specific point. Otherwise without the point number, the program will use the next point with the current code. The "3RD" code is used to specify the third symbol insertion point and similar to the "2ND" code, a point number after the "3RD" is optional. The "2ND" and "3RD" codes should be assigned to the first point. For example, consider a code of "CAR" with a three point symbol insertion definition. If point #1 has a description of "CAR 2ND 3RD", then point #1 will be used as the first symbol insertion point and the next two points with the "CAR" description will be used as the second and third symbol insertion points.



Multi Point Symbol Drawing

Custom Attributes: This feature allows you to use customized blocks that have customized attributes (the tag/value pairs). This feature works for both point attribute blocks and symbols. For attribute blocks, Field-to-Finish looks for attributes with the tags "PT#", "ELEV2", and "DESC2". The custom attributes feature allows you to define up to an additional 5 attributes in their custom blocks on a per-field code basis. For example, the custom block could have an attribute with the tag "TREE_SPECIES" and there's a separate field code for each species of tree. Each of those field codes can specify the value that should be assigned to the attribute that has the TREE_SPECIES tag. Then when the points are drawn, the tree species is shown. Note that the custom attributes must have their Constant and Preset properties set to "no". The custom attributes settings in F2F should not use those tags that the software already handles (PT#, ELEV2, and DESC2), or the setting will be ignored.



Set Color: The line work will be drawn in this color. The default is BYLAYER.

Text Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size.

Symbol Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size in AutoCAD. The horizontal scale can be set in *Drawing Setup*.

Line Width: This controls the width for the linework. Only applies to 2D polylines.

Line Type Spacing Scaler: This is a scaler value that is multiplied by the AutoCAD LTSCALE system variable to give the distance between symbols in the line.

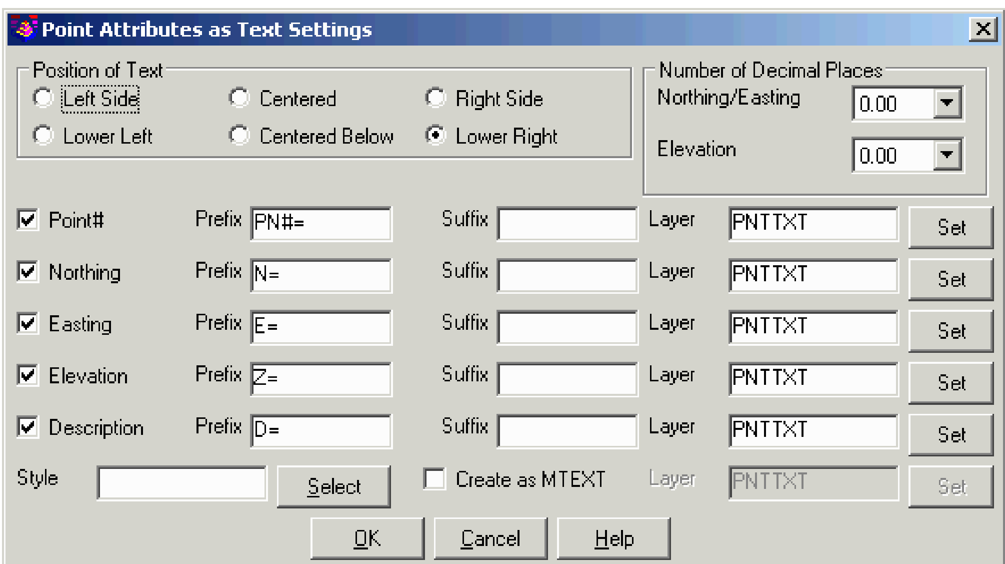
Line Type Text Scaler: This is a scaler value that is multiplied by the AutoCAD LTSCALE system variable to give the size of the text in a line.

Unit Symbol: This option will draw the point symbol at unit (1:1) scale. For example, this option could be used for a symbol that is already drawn to actual dimensions such as a car symbol.

Set Template: For 3D polyline codes, this option allows you to assign a template (.TPL) file to the code. The code points act as the centerline for the template and the program will drawn parallel 3D polylines for each break point (grade ID) in the template. The template file is defined in the Civil Design module.

Entity Type: This defines the line entity to be created. Points only does not create any line work. 3D Polyline can be used for breaklines. 3D and 2D entity type selection creates a 3d polyline in the layer specified in the Dual 3d polyline layer setting and a 2d polyline in the layer identified in the Layer setting. Since 3d polylines do not display linetypes, this is useful when needing linework in 3d for design work while also needing to display linetypes for final plotting of the drawing. This provides an easy and quick way to turn off all 2d polylines or all 3d polylines by using the layer control dialog or the appropriate toggles in the Draw Points dialog.

Attribute Format: This chooses the type of point entities to create. The Attribute Block format creates the Carlson point entity which is block with attributes for point#, elevation and description. The Text Attribute format creates text entities for each of the point attributes. When the Text Attribute format is selected, the Set button is available where you can control which attributes to draw as text and the position, decimals, style, prefix, suffix and layer for each attribute.



Separate Attribute Layers: This controls the layers of the point and symbol attributes. With "None" the point layers are the standard layers, "PNTNO", "PNTELEV" and "PNTDESC", and the symbol layer is "PNTMARK". With "Points" or "Both" the point attribute layers begin with the layer for the code followed by the attribute type. For example, the "DWL" code shown in this dialog has a layer name "DRIVEWAY". The point attributes would then be "DRIVEWAYNO", "DRIVEWAYELEV" and "DRIVEWAYDESC". With "Symbols" or "Both" the symbol attribute layer begins with the layer for the code followed by "MARK".

Hard Breakline: This will tag the 3D polylines created with this code as hard breaklines. In *Triangulate & Contour*, contours are not smoothed as they cross hard barriers.

Smooth Polyline: This applies a modified Bezier smoothing to the polyline. The smoothed polyline will pass through all the original points.

Connection Order: The points of a distinct code can be connected in their point number order or by nearest found which makes the line by adding the next closest point.

Tie: When checked the linework drawn with this code will always close. For example if you have points 1, 2, 3, and 4 with the code BLDG and Tie is checked on for the code BLDG, then the linework will be drawn from point 1 to 2 to 3 to 4 and then back to point 1, closing the figure.

Elevation Integers: This controls the number of digits to display to the left of the decimal point for the elevation label. The All setting will show the full elevation digits. The other settings allow you to limit the number of digits to display for the purpose of reducing the amount of space the elevation labels take up in the drawing. For example, if a site is in the 4000 foot elevation range, then this setting could be set to three digits (000) and an elevation of 4321 would be labeled as 321.

Elevation Decimals: This controls the display precision for the elevation label.

Elevation Prefix/Suffix: These set the prefix and suffix for the elevation label per code. In the Draw function under Additional Draw Settings, there is an override to set the elevation prefix/suffix for all the codes.

Attribute Layout ID: Controls the location of the point number, elevation and description. These attribute layouts are defined in AutoCAD drawings that are stored in the Carlson SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing.

Locate Pts on Real Z Axis: This option will draw the points at the actual point elevation. Otherwise the points are drawn at zero elevation. For example, you could turn this option off for the FH for fire hydrant code to drawn them at zero. Then the GND code could have this option on to draw the ground shots at their elevations.

Random Rotate: This option will randomly rotate the symbol. For example, this option could be used for tree symbols to have the trees drawn in various orientations.

Random to Line: This option applies to points that are part of Field-to-Finish linework. This option will align the point attributes and symbol to the associated linework.

Distinct Point Layer: When this toggle is selected, the line work is created in the layer defined in the Layer field and the points are created in the specified distinct point layer. For example, you could have DRIVEWAY for linework and DRIVEWAY_PNT for the points.

Code Definitions (continued)

Select All: This option selects all the codes. This can be used when only wanting to process a couple of codes. For example, use the select all option to select all the codes and then turn them off. Now select the codes for processing and turn them on. Also it can be used to make a global change to all the codes.

Add: The new code definition is inserted in the list in the position after the currently selected one. If none are selected for positioning, the new code is placed at the top. Only one code definition may be highlighted before running this routine.

Copy: This option copies the definition of a selected code. It opens the Edit Field Code Definition dialog and copies the definition of the selected code to the appropriate settings. It does not copy the name of the code. It is a time saving tool to use when creating codes that are similar with only a couple of differences.

Cut: This command will remove the highlighted code definitions from the list and puts them in a buffer for retrieval with Paste.

Paste: This command will insert the code definitions put in the buffer by the Cut command. These codes will be inserted after the row of the currently highlighted code or at the top.

Search: Allows you to search for a specific code in the list.

Coordinate File

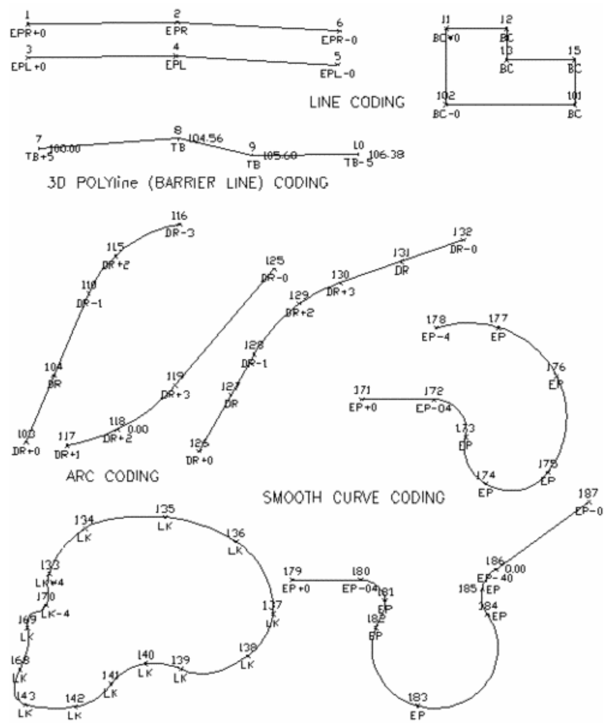
Set CRD File: This command allows you to specify a coordinate (.CRD,.CGC,.MDB,.ZAK) file to process.

Edit Points: This command opens the *Edit Points* spreadsheet editor. See *Edit Points* for more details.

Draw: This command returns to the Draw Field to Finish dialog box.

PointCAD Coding

Field-to-Finish supports an early Carlson style of linework coding called PointCAD. The PointCAD codes use numbers with +,-,∗ symbols as follows:
+0 Starts a regular 2D line (not a polyline) that is open.
∗0 Starts a regular 2D line that is closed.
+4 Starts a curved 2D polyline that is open.
∗4 Starts a curved 2D polyline that is closed.
+1 Begins a 3-point arc.
-0 or -1 or -3 or -4 or -5 or -6 or -7 Ends a line.
+5 Starts a 3D polyline that is open.
∗5 Starts a 3D polyline that is closed.
+6 Starts a 2D polyline that is open.
∗6 Starts a 2D polyline that is closed.
+7 starts line whose type (2D line, 2D polyline, 3D polyline) is specified by the point's field code definition. If the field code definition is to use points, then a 2D line is started.
+2 Middle point of 3 point arc
-05 starts a curved 3D polyline section.
-50 ends a curved 3D polyline section.
+8 starts a 2D and 3D polyline combination that is open.
∗8 starts a 2D and 3D polyline combination that is closed.
-8 ends a 2D and 3D polyline combination.
-08 starts a 2D and 3D polyline combination curve that is open.
-80 reverts back to a straight 2D and 3D polyline combination.



PointCAD linework coding examples

Pulldown Menu Location: Survey
Keyboard Command: fld2fin
Prerequisite: A data file of points with descriptions
File Names: \lsp\finish.arx, \lsp\field.dcl, \lsp\deffield.lsp

Spreadsheet Edit Points

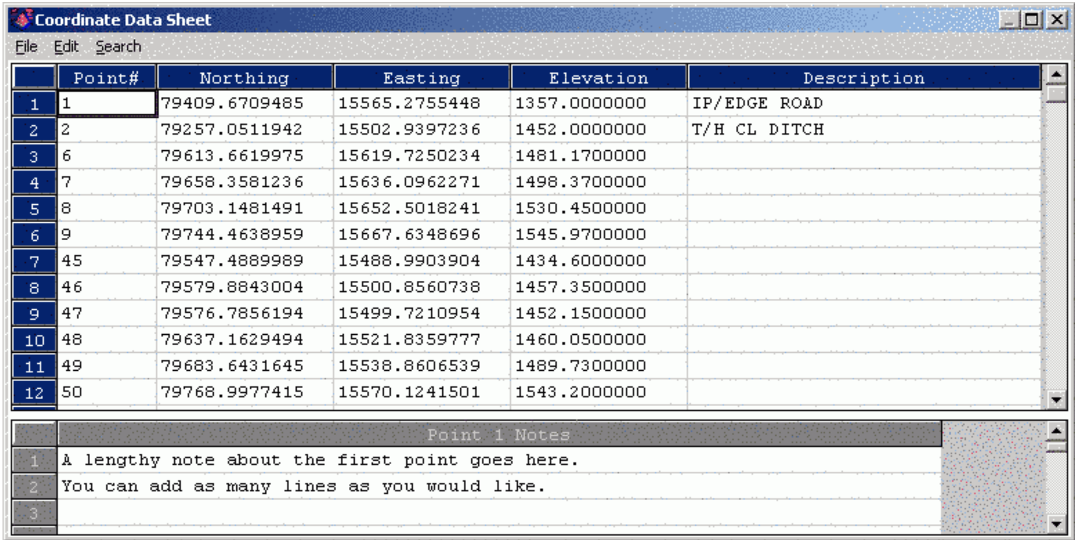
Function

This command edits point data in the current coordinate file. The current coordinate file can be set with the Set Coordinate File command. Edit Points shows all the points in the coordinate file. New points can be added and points can be deleted by using the Insert and Delete keys.

This tool also lets you edit notes associated with each point. While the standard point description is limited to 32 characters, the drawing notes are not. When you click on a given point, you can add numerous lines of notes about that point in the bottom of the dialog. Keep in mind that these notes are stored in a separate file with the extension ".not" having the same name as the CRD and residing in the same folder.

Keyboard Command: EDITPT

Prerequisite: None



On-Screen Edit Points

Function

This command will edit the attributes of a TakeOff point such as the symbol type, point number, elevation and description. When this command is invoked, the command line will prompt the user: **Select point to edit (Enter to end)**. At this point, you can select any part of the point including the symbol, elevation, point number or the description. Next, a dialog will appear as shown.

To change the symbol, either type in a new symbol name in the edit box, or choose the "Select Symbol" button where you can choose from a list of symbols. To change any of the other properties of the point, simply change or replace the contents of the edit box with the new information. If you change the point number to a number that already exists in the current CRD file, and point protect is ON, you will be prompted **[O]verwrite w/new coordinates, overwrite [A]ll, or use number <1000>:.** You can choose to use the next available point number in the CRD file (this is the default) or overwrite the point number. The properties that you modify with this command will update the current CRD file and the screen entities.

You may also choose to use the AutoCAD *DDATTE* command to change the attributes of a point. If you do this, then the CRD file will not be updated and if you change the elevation attribute, the point will not change its current Z location.

Prerequisite: TakeOff points

Keyboard Command: EDITPNT

Scale Point Attributes

This command will scale point attribute text (number, elevation and descriptions) and point symbols up or down in size. The routine prompts for a scale multiplier and a selection set of objects. If you want to enlarge, enter a value greater than one. If you want to reduce, enter a decimal fraction such as .5. This would reduce the text size by 50%. This command is very useful if you have set up your drawing for one plotting scale and decide to change to a

new plotting scale. This command has the added benefit that it will adjust the point attributes and symbols to a new screen twist angle.

Prompts

Scaling Multiplier <0.500>: *2.5* This response would enlarge the point attributes and symbols by 250 percent.
Scale symbols only, point labels only or both [Symbols/Labels/<Both>]? *press Enter*
Select points from screen, group or by point number [<Screen>/Group/Number]? *press Enter*
Select Carlson Software points. *pick a point*
Select objects: Specify opposite corner: *pick a point*
Scaling Carlson Software Point Attributes ....
Number of entities changed> 174

Pulldown Menu Location: Points
Keyboard Command: pntenl
Prerequisite: Carlson points
File Name: \lsp\pntenl.lsp

Resize Point Attributes

This command sets the size of the selected point attributes (point number, elevation, description) and point symbols. This command is similar to Scale Point Attributes, but instead of scaling the size by a factor, all the select points are set to the same specified size. Points can also be chosen based upon Point Groups.

Prompts

Enter point attribute and symbol size <4.0>: *press Enter*
Scale symbols only, point labels only or both [Symbols/Labels/<Both>]? *press Enter*
Select points from screen, group or by point number [<Screen>/Group/Number]? *press Enter*
Select Carlson Software points.
Select objects: *pick the point entities*
Finding Carlson Software Point Attributes ....
Number of entities changed> 10

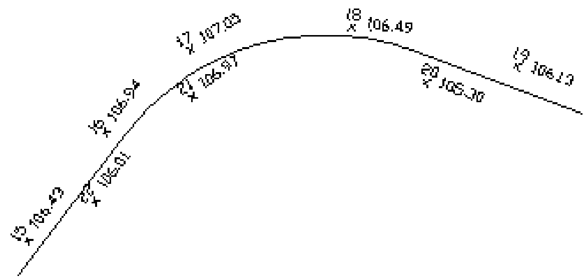
Pulldown Menu Location: Points
Keyboard Command: sizepnt
Prerequisite: Carlson points
File Name: \lsp\sizepnt.lsp

Twist Point Attributes

This command will rotate the orientation of the text of Carlson point attributes (point #, elevation, description) and point symbols. The Twist Screen option aligns the point attributes to appear horizontal in the current twist screen. The Azimuth option allows you to enter an azimuth or pick two points to align the point attributes. The Entity Segment option aligns the point attributes by the selected line or polyline segment in the direction the entity is drawn. The Follow Polyline option aligns the point attributes by the polyline segment that is closest to the point.

Prompts

Twist by [<Twist screen>/Azimuth/Entity segment/Follow polyline]? *F*
Select reference polylines to follow. *pick a polyline*
Select objects: 1 found
Select objects:
Select points from screen, group or by point number [<Screen>/Group/Number]? *select Enter*
Select Carlson Software points.
Select objects: *pick the Carlson point inserts*



Point attributes aligned by Follow Polyline option of Twist Point

Pulldown Menu Location: Points
Keyboard Command: twistpts
Prerequisite: None
File Name: \lsp\twist_pt.lsp

Erase Points

This command erases Carlson points inserts from the drawing. The points to erase can either be selected from the screen or specified by point number, point number range or by point group. Erasing a Carlson point will erase the point symbol, point attributes, and point node. The points may optionally be erased from the coordinate file. As long as the points are not deleted from the coordinate file, they can be redrawn with *Draw-Locate Points.*

Prompts

Select points from screen, group or by point number [Screen/Group/<Number>]? *press Enter*
Point numbers to erase: *1-5*
Delete points from coordinate file (Yes/<No>)? *press Enter*
Erasing Carlson Points ....
Number of points erased> 5

Pulldown Menu Location: Points
Keyboard Command: DELPT
Prerequisite: Carlson points to be erased
File Name: \lsp\delpt.lsp

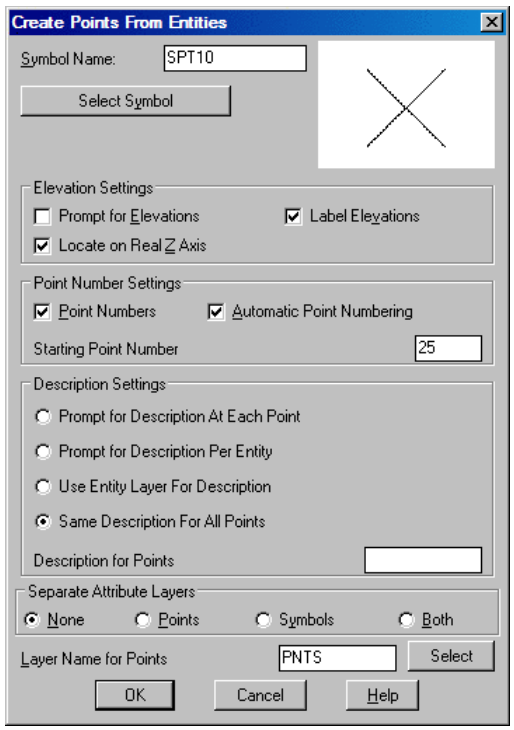
Create Points from Entities

This command will create Carlson points on selected entities. The points are stored in the current coordinate (.CRD) file and drawn on the screen. For arcs and polylines with arc segments, points are created at the radius points of the

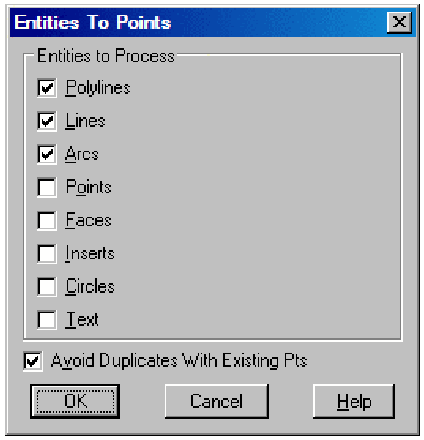
arcs as well as the PC and PT.

In the first options dialog, there are settings for the point attributes. To have points obtain their elevation from the selected entities, unselect the **Prompt for Elevations** toggle and select the **Locate on Real Z Axis** toggle. After you have specified the point options, a secondary dialog appears which allows you to specify the entity types to process. Under the **Description Settings, Prompt for Description At Each Point** will prompt you at the command line for a description for each individual point. **Prompt Per Entity** will ask you for a description per each highlighted entity. **Use Entity Layer for Description** will assign the layer name to the description. When Entity Layer for Description is checked, the layer name of the entity will be used as the description for the created point. **Same Description For All Points** will prompt you for a single description for all points.

The second options dialog has processing settings. When **Avoid Duplicates with Existing Pts** is checked, this routine will not create a point if a point with the same coordinates already exists in the current coordinate (.CRD) file.



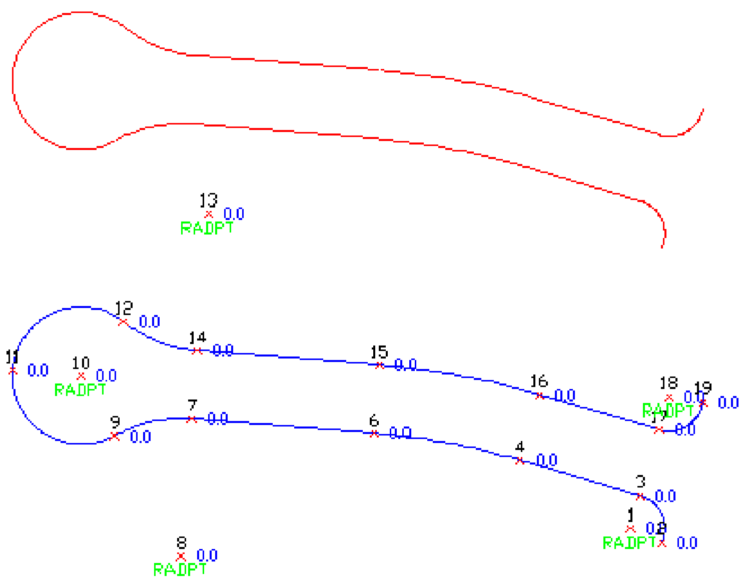
Routine begins with this dialog



After clicking *OK* on the first dialog

Prompts

Create Points From Entities Dialogs Choose settings
Select arcs, circles, faces, points, text, lines and polylines.
Select objects: *pick entities*

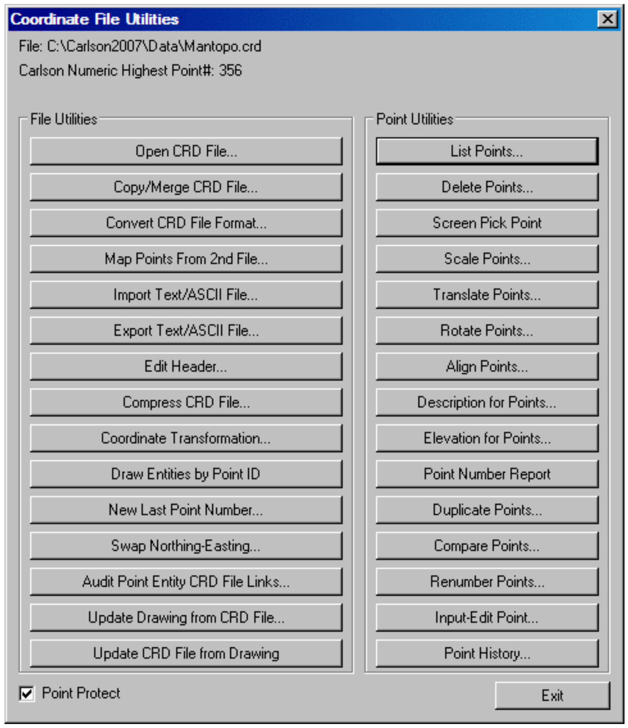


Before and after using Create Points from Entities. Points are created at each endpoint and radius point.

Pulldown Menu Location: COGO
Keyboard Command: autopnts
Prerequisite: drawing entities
File Names: \lsp\crdutil.arx, \lsp\autopnts.lsp

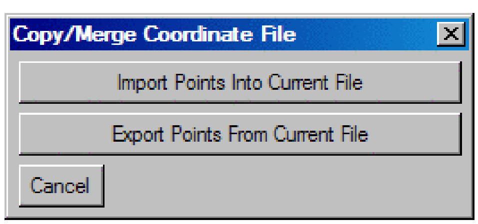
CooRDinate File Utilities

This command allows you to manipulate the coordinates stored in a coordinate (.CRD) file. One of the most important commands is the Update CRD File from Drawing which allows you to update the file after editing the drawing with commands such as *Erase*, *Move*, *Rotate* or *Change Elevations*. Another handy option is the *Draw Entities by Point Number* which allows the user to input point number ranges and plot Lines, Arcs, Polylines or 3D polylines. Coordinate files have either numeric or alphanumeric point numbers. Alphanumeric point numbers consist of nine or less digits and letters (i.e. point number 7A). The type of point number format is displayed at the top title bar of the main dialog.

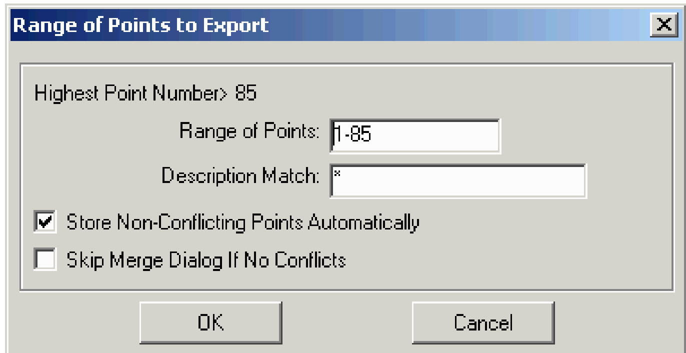


Open CRD File: Allows the user to switch to another file. When you exit Coordinate File Utilities this will be the current file that you work with in Carlson.

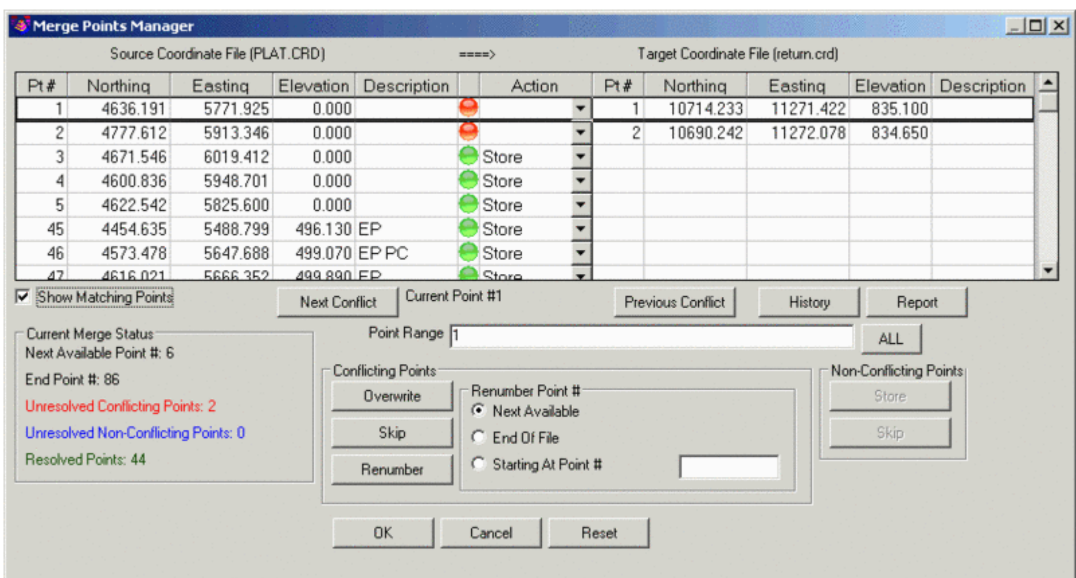
Copy/Merge CRD File: This command allows for the copying of entire CRD files, or parts of CRD files, to a new or existing files. This can be used to make a backup of your coordinate file, and it can also be very valuable in coordinate file manipulation. For example, if a certain range of points from one CRD file was also required in the active CRD file, this command would be used to simply copy the required range into the active CRD file. There are two options when first executing the command. These options are whether to import points from another file to the current (active) CRD file, or to export the current (active) coordinate file to another file.



Once this option has been decided, a prompt for the file to copy From or TO, will be displayed. Here simply specify the correct file.



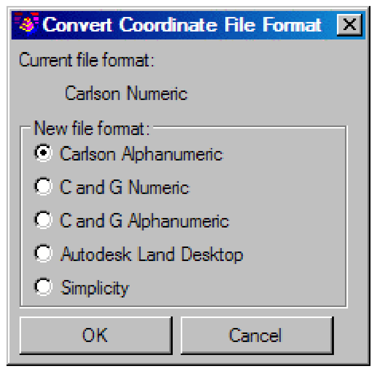
Next there's a dialog to specify the range of points to transfer and some options. Here specify the points to copy. Point numbers and ranges can be entered together, for example, 1-3,10,15 would result in points 1 through 3 and points 10 and 15 being copied. The Description Match can be used to filter the points to transfer only the points with matching description. The default of * will trasnfer all the points in the range. The Store Non-Conflicting Point Automatically will set the transfer action as Store for all transfer points that don't have a point protect conflict. The Skip Merge Dialog If No Conflicts will skip the next dialog when there are no point protect conflicts.



Next there's the Merge Points Manager dialog that shows the Source Coordinate File on the left (where the point data is being copied from) and the Target Coordinate File on the right (where the point data is being written to). Conflict cases are when the same point number exists in both files with different coordinates. The action choices for conflicts are to Overwrite, Skip or Renumber. For renumber, you can either renumber with the next available point number in the target file or to the highest point number in the target file plus one. Non-conflict cases are when the source point number does not exist in the target file. The action choices for non-conflicts are to Store or Skip.

You can assign actions by picking on the Action field in the spreadsheet or by entering in a Point Range to apply and picking an action button. The Show Matching Points toggle will show points with matching point data in both files. Otherwise only point with differences are shown. The Next Conflict button will highlight the spreadsheet and set the Point Range to the next point that needs an action assigned. Similarily, the Previous Conflict sets focus to a lower point number that needs an action. The History button shows the point history for the selected point. The Report button creates a list points report. The Current Merge Status reports the number of unresolved and resolved points. When all the unresolved points are resolved by assigning actions, you can pick OK.

Convert CRD File Format: This allows you to convert the current CRD file from numeric format to alphanumeric format or visa versa. This routine will also change crd files to and from different software formats. These formats include C&G, AutoDesk Land Desktop, and Simplicity. The current format of the active coordinate file will be displayed as well as the options for the new file format. This command only changes the format of the active coordinate file.

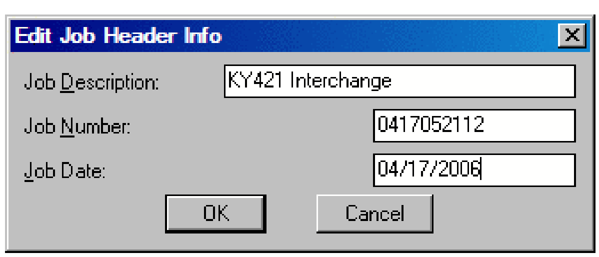


Map Points from 2nd File: This routine adds point to the current CRD file from points stored in a second CRD file. The points to copy are specified by numbers one at a time. Prompts for the destination point number (number to create in current crd file) and source point number (point number to be copied from second crd file) will be displayed.

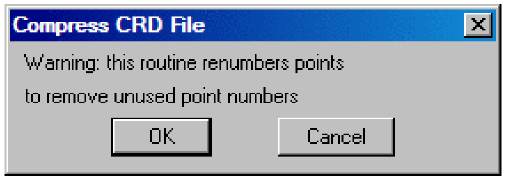
Import Text/ASCII File: This routine converts point data from a text file into the current coordinate (.CRD) file. See the *Import Text/ASCII File* command in this chapter for more information.

Export Text/ASCII Text File: This routine outputs point data from the current coordinate (.CRD) file to a ASCII Text file. See the *Export Text/ASCII File* command in this chapter for more information.

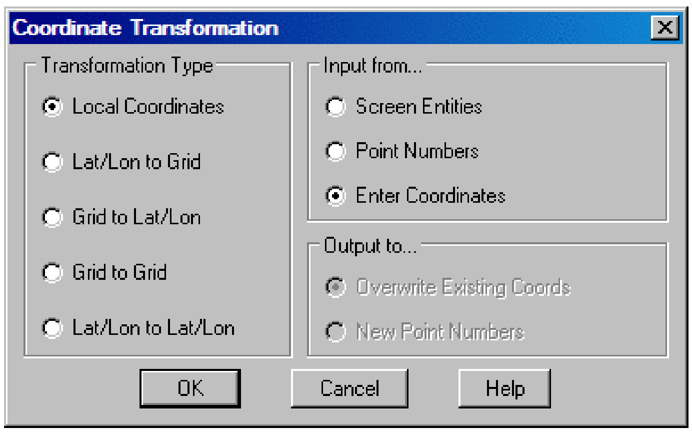
Edit Header: Enter or edit the job information associated with the coordinate file. The fields include Job Description, Job Number and Job Date. This information will appear on the List Point report. Non-digit characters are not allowed in the Job Number field.



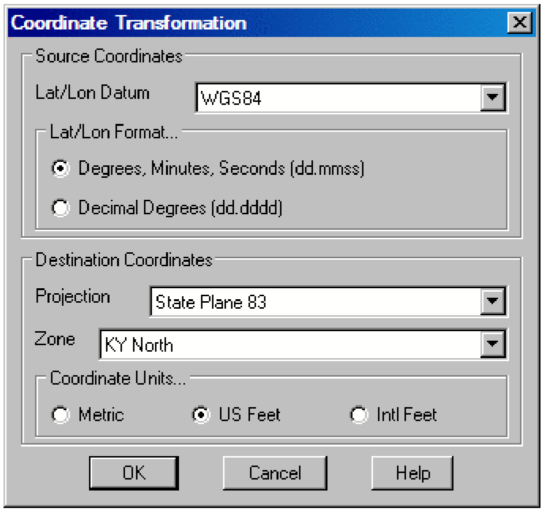
Compress CRD File: Removes unused point numbers by renumbering high point numbers into the unused spaces. For example, for an original file with points 1,2,105,107,108,109 would be compressed to 1,2,3,4,5,6.



Coordinate Transformation: Transforms coordinates between local, state plane 27, state plane 83, latitude/longitude, and Universal Transverse Mercator (UTM). Works on individually entered coordinates, by range of point numbers and with on-screen entities. For converting between state plane 27 and 83, Carlson calls upon NADCON from the National Geodetic Survey to apply the latitude/longitude adjustment. The NADCON program, ndcon210.exe, is stored in the Carlson EXEC directory.

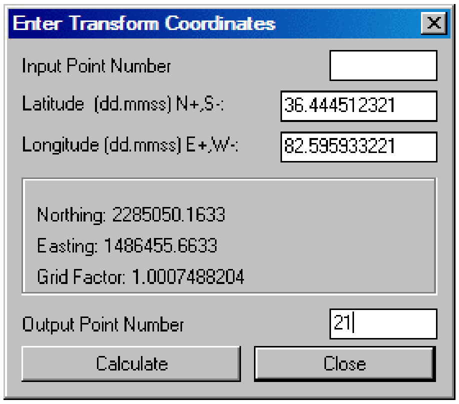


The Transformation Type is used to define the Source Coordinate and Destination Coordinate formats. Settings for Lat/Long Datum, Lat/Long formats (dd.mmss or dd.dddd), Projections, State Plane Zones and coordinate units are defined in the Transformation Type dialog. The format of this dialog will change depending upon the type of transformation requested.



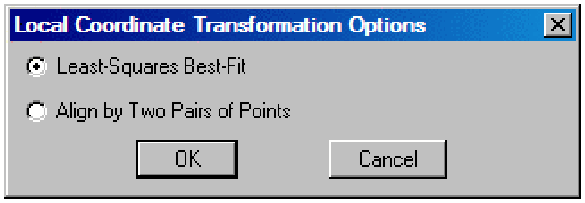
Example Lat/Long to Grid dialog

For all Transformation types, there are three options for inputting the data to be transformed. Data can be selected from the screen by using the **Screen Entities**. If a range of points or a particular point is desired, the **Point Numbers** option would be used. Manual entry of coordinates to transform one at a time is accomplished with the **Enter Coordinates** option. The coordinates can be typed in or use the Input Point Number option. Output Point Number is an option to store the results in the coordinate file.

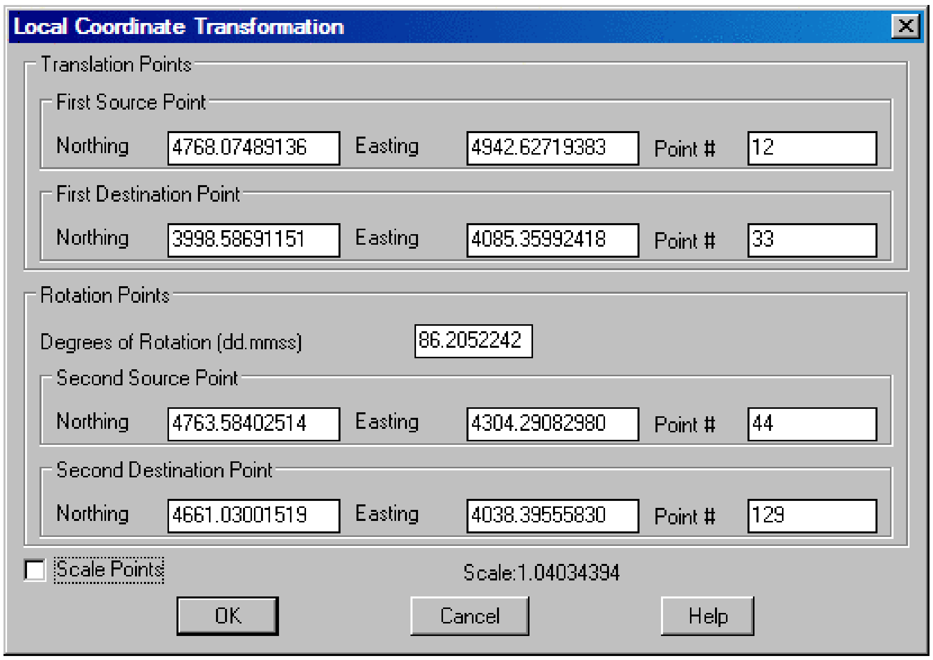


For all transformations there are two output options when using point numbers as the input data. **Overwrite Existing Coords** replaces the original coordinate values with the new coordinate values after transformation. **New Point Numbers** will retain the original coordinate data and point numbers and create new point numbers with the revised coordinate data after transformation.

When transforming a **Local Coordinate System**, there are two options for defining the transformation as shown in the next dialog.

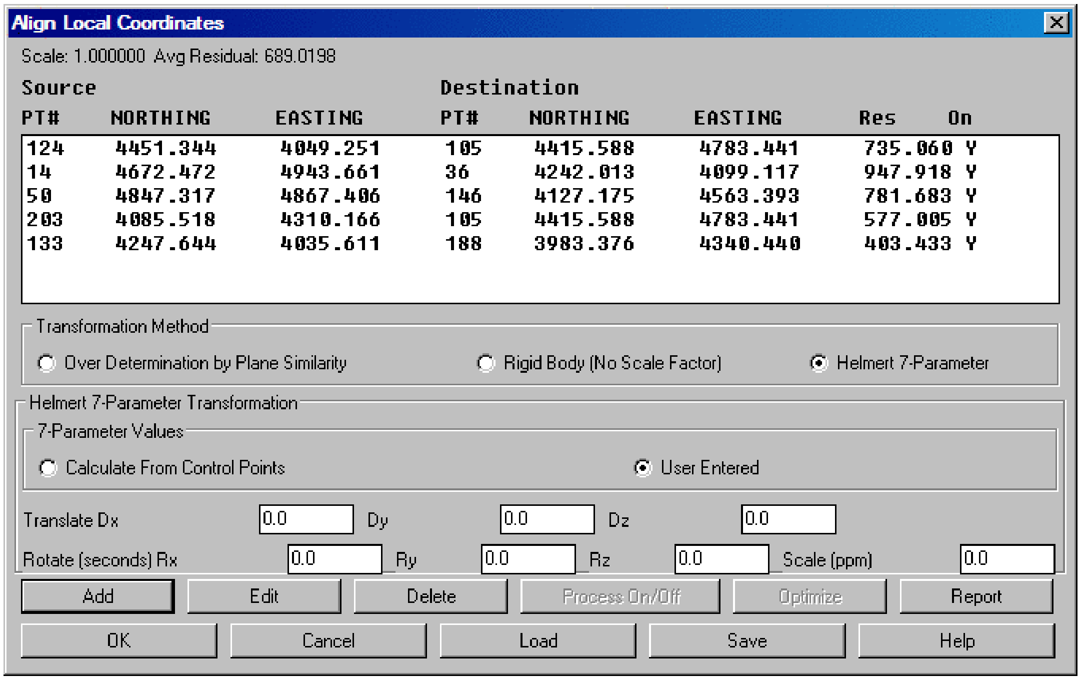


The **Align by Two Pairs of Points** option uses two pairs of source and destination coordinates. The first pair defines the translation as the difference between the source and destination northing and easting.



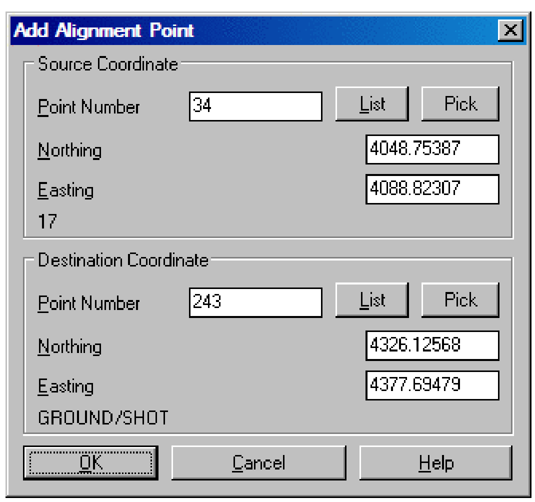
This destination point is also the pivot point for rotation. Rotation can be entered directly or defined by a second pair of points where the bearing between the first and second source points is rotated to align with the bearing from the first and second destination points. There is an option to also apply scaling. The scaling holds the angle between points and adjusts the distances by the scale factor. The scale factor is calculated for each point as the elevation factor at the first source point times the grid factor at the first destination point averaged with the elevation factor at the transform point times the grid factor at the transform point.

The **Least-Squares Best-Fit** option is used when there are more than two pairs for translation points. Since two pairs of points are sufficient to define the translation and rotation, more than two pairs of points provides more than enough information.



Over **Determination by Plane Similarity** is used to find the least squares best fit transformation for all the given source and destination points. Besides doing a translation and rotation, this option will also scales the points during the transformation. **The Rigid Body Transformation** also does a best fit least squares transformation, but applies only translation and rotation with no scale. The **Helmert 7-Parameter** method can also be used for local transformations. The **7-Parameter Values** can be calculated from control points or entered by the user.

The **Add** button is used to define the source and destination coordinates for the points that define the transformation. Pressing this button brings up the following dialog box.



The **Edit** button is used to edit existing data.

The **Delete** button removes the source and destination pairing from the transformation setup.

The **Process On/Off** button allows source and destination pairings to be turned on and off. This is useful when wanting to inspect different results using different pairings.

The **Optimize** option chooses which point pairings would yield the best transformation results by turning off the processing of pairings with higher residuals. This minimizes the average residual for the control points.

The **Report** option displays a report of the transformation point pairings, their residuals, processing status, transformation scale and avg. residual.

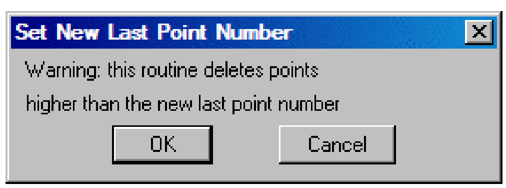
The **Load** and **Save** options allow for saving and recalling local coordinate transformation pairings and settings.

Draw Entities by Point ID: Draw Lines, Arcs, 3DLines, Polylines or 3DPolys by defining a range of point numbers.

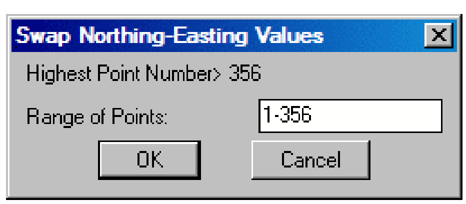
Prompts

Plot Entities by Point Number
Type of entity, Arc/Polyline/3dpoly/2dline/Exit/<Line>: *P* This response causes the program to plot polylines.
Example: '1*4-7-10*12-5-8' would draw lines from point number's 1 through 4 then to 7, to 10 through 12, then to 5 to 8. (limit 132 characters)
Undo/<Enter point numbers or ranges>: *1*10-20*30*
The program draws a polyline from point number 1 through 10 to point number 20 through 30.

New Last Point Number: This option sets the highest point number in the CRD file. All points above this number are erased.

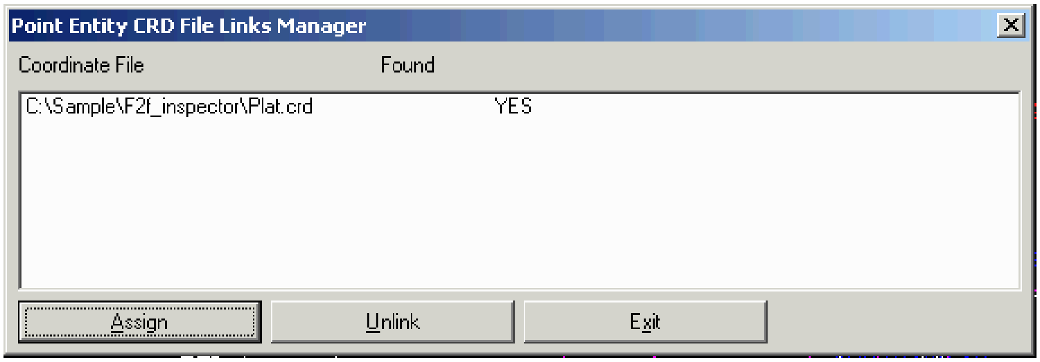


Swap Northing-Easting: This option allows you to swap northing and easting coordinates for any selected range of points. What was the northing of an existing coordinate point, or range of points, becomes the easting. And the easting(s) becomes the northing(s).

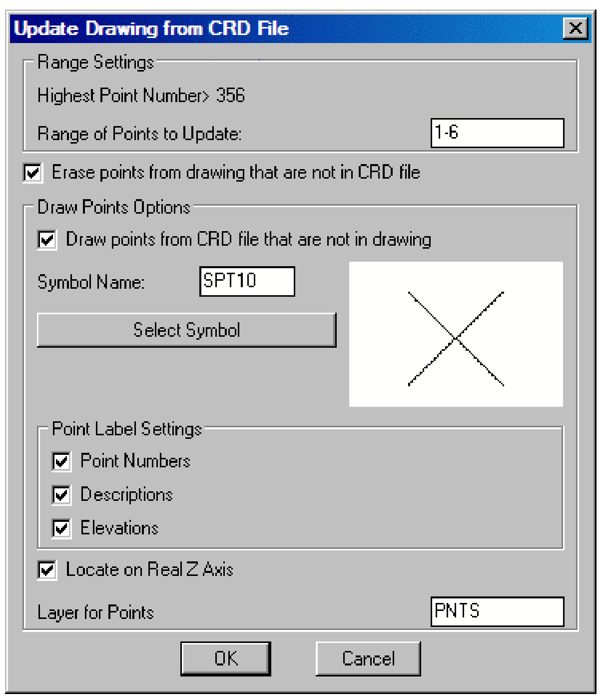


Point Entry CRD File Links Manager: When points are created in the drawing, the program records the source coordinate file for the points. The coordinate file names assigned to the point entities links the point entities back to the coordinate file. These links are used by routines that process the point entities and then need to reference the coordinate file such as Move Point which selects a point entity and updates the coordinate file. This routine checks all the point entities in the drawing and lists all the linked coordinate files. You can use the Assign button to set the

coordinate file assigned to point entities which is useful when the coordinate file has been moved after the points were drawn. Use the Unlink button to remove the link.



Update Drawing from CRD File: This function updates the position of Carlson points in the drawing to match the position stored in the coordinate file. This command also has options to erase and draw points. For the erase option, points are erased from the drawing if the point number does not exist in the coordinate file. For the draw option, if a point number in the CRD file does not exist in the drawing, then this point is drawn using the settings from the dialog. The number of points modified, erased and drawn is reported at the end of the command.



Update CRD File from Drawing: This function allows you to select all or some of the points in the drawing and add or update them to the .CRD file. The points can be filtered with AutoCAD's Select Objects: selection mechanism and/or wild card matching of the point descriptions. The Update Point Descriptions option determines whether the point descriptions from the drawing will be stored to the CRD file. Use this command to update the file after a global edit such as *Move, Rotate, Renumber Points, Change Elevations, Erase,* etc. This routine directly reads Leica

(Wildsoft), Softdesk, Geodimeter, InRoads, Land Development Desktop, and Eagle Point point blocks.

List Points: List the points stored in the .CRD file. See the *List Points* command in this chapter for more information.

Delete Points: Deletes points in the coordinate (crd) file by point number or description.

Screen Pick Point: Pick a point on the graphics screen and it's coordinate values are added to the coordinate (crd) file. Prompts for point number, elevation and description will be displayed. This command does not plot a point, point attributes or point symbol. Use the command *Draw-Locate Points* command to do this.

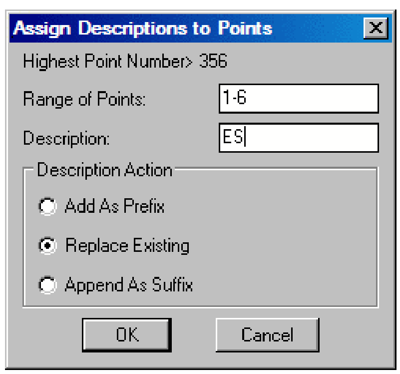
Scale Points: This option multiplies the point northing, easting, and elevation by the scale conversion factor. You can use this routine for metric-English conversion. See the *Scale Points* command in this chapter for more information.

Translate Points: This option translates a range of points based on entered delta x and delta y, entered coordinates or translation point numbers. See the *Translate Points* command in this chapter for more information.

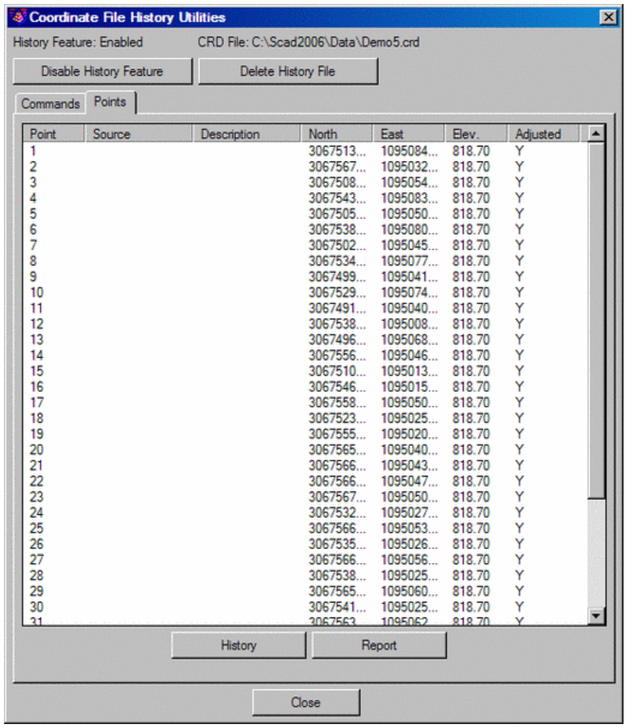
Rotate Points: This option rotates a range of points based on entered degrees or rotation, entered azimuths, entered bearings or rotation point numbers. See the *Rotate Points* command in this chapter for more information.

Align Points: This option does a translate based on a source point and destination point and then rotates to align the first source point and a second source point with the first destination point and a second destination point. See the *Align Points* command in this chapter for more information.

Description for Points: This option sets the point description field with the user-specified text for a range of point numbers.

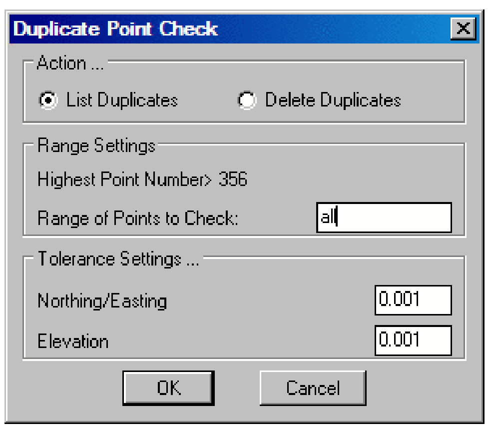


Elevation for Points: This option sets the elevation of a specified point or range of points.

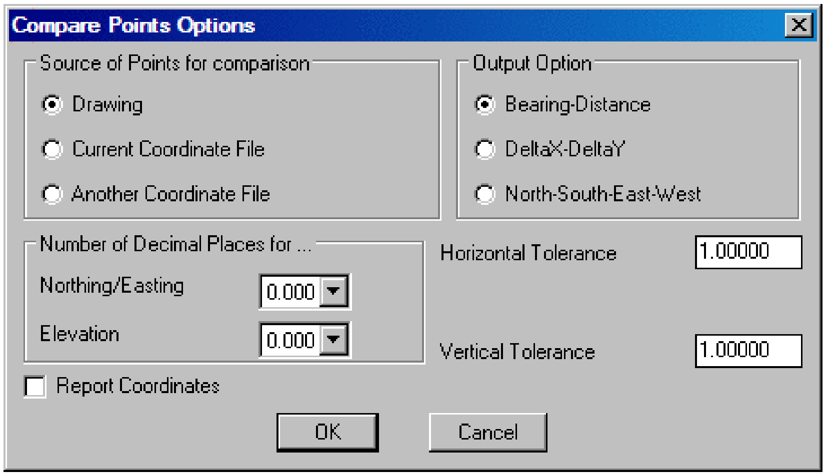


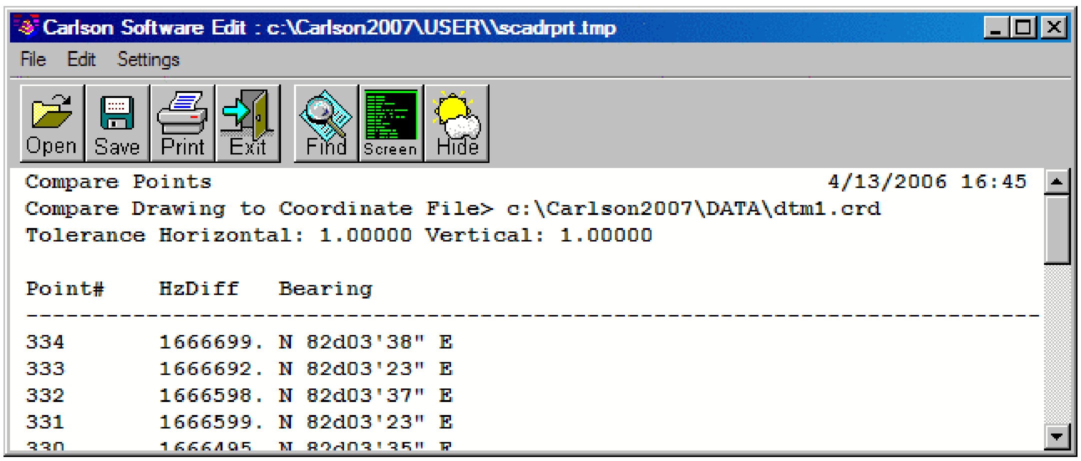
Point Number Report: This routine lists the used and the unused point numbers in the CRD file.

Duplicate Points: This function searches the CRD file for points with the same northing, easting and elevation. The tolerances for considering points to have the same coordinate are set in the dialog separately for northing/easting and elevation. To be counted the same coordinate, both the northing/easting and elevation must be within the tolerance distance. The duplicate points can be erased or only reported. For the erase option, the first point number is kept and any higher point numbers with duplicate coordinates are erased from the CRD file.



Compare Points: This function compares the coordinates in the .CRD file with either the coordinates for the matching point numbers in the drawing file, with matching point numbers from another CRD file or with different point numbers from the same CRD file. A report is created for any differences that shows the point numbers and the differences. The difference can be reported as a bearing and distance between the two points, as distance North/South and East/West or as the delta-X and delta-Y. There is an option whether to include the point coordinates in the report.

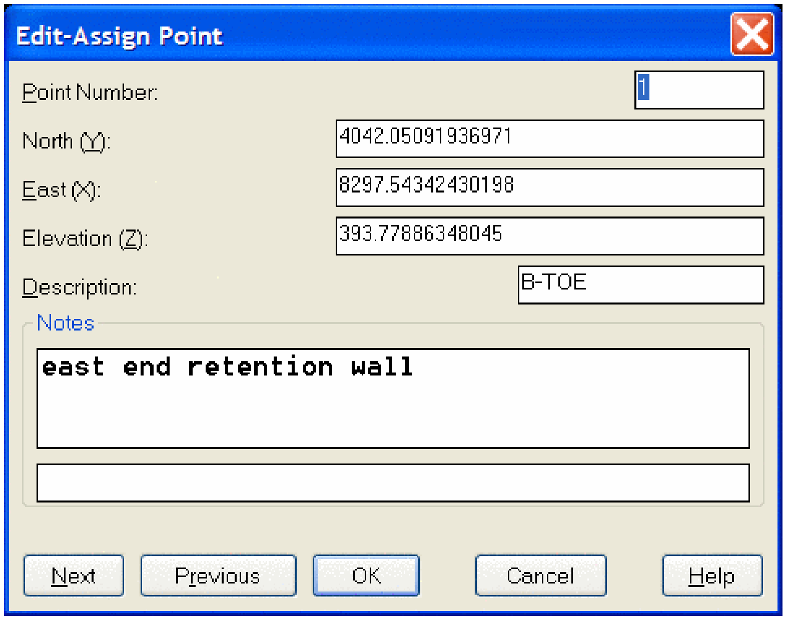




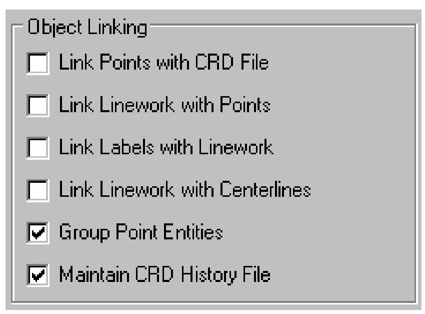
Example Bearing-Distance format Compare Points Report

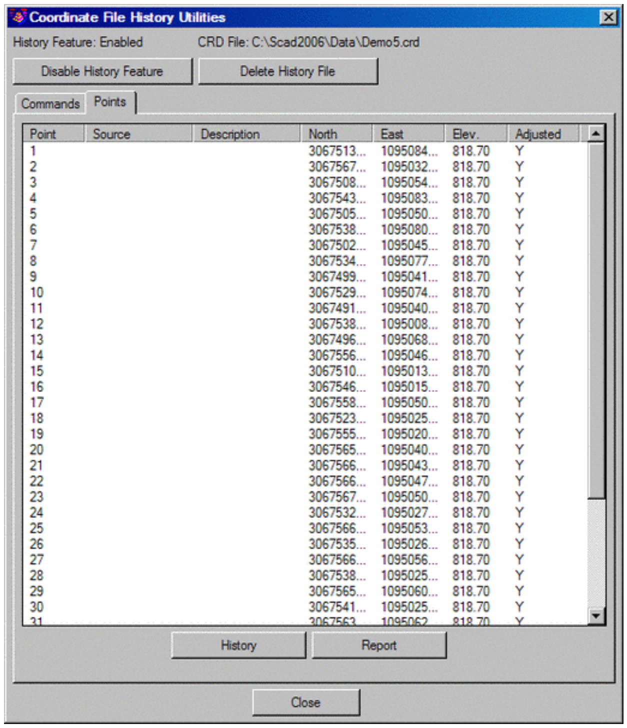
Renumber Points: This option renumbers points in the user-specified range starting from a new point number. The old point numbers are erased. The condense points will renumber such that there are no unused point numbers in the renumbered range. Otherwise the spaces between the points is maintained. In the example shown, renumbering 1-25 with points 1,2,24,25 to starting point number 101 will result in points 101,102,103,104 if condense is on or 101,102,124,125 if condense is off.

Input-Edit Point: Enter or edit the coordinate values or the description of a point. The Notes section is for adding optional point notes which are additional point descriptions. The standard description field is limited to 32 characters. Under notes, any number of lines of text can be assigned to the point. A list box shows the lines of notes. To add a note line, pick a blank line in the list box and then type in the note in the edit box belong the list box and press Enter. To edit a note, highlight the line in the list box and edit the text in the edit box.



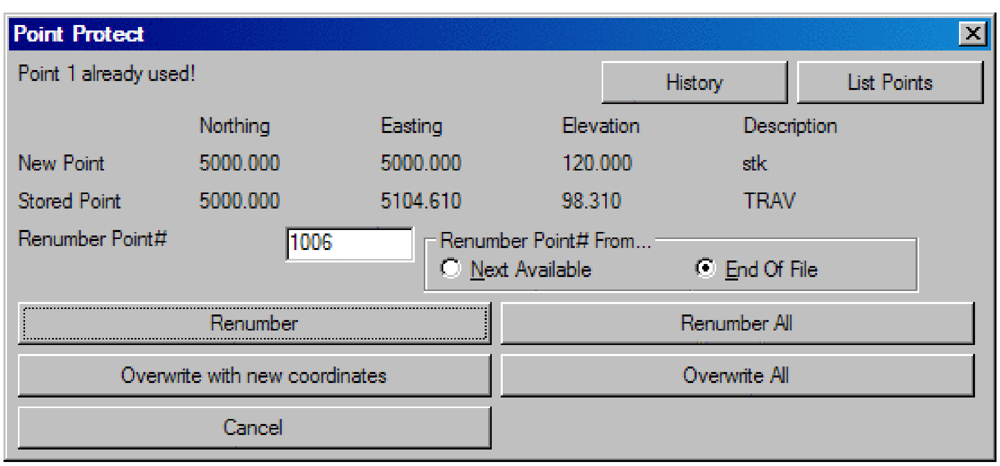
Point History: All changes to the coordinate file will record the commands performed on this coordinate file and the status of the points themselves. This makes up the coordinate file history. The history can then be reported by point number or by command. All of the changes can be rolled back. It is important to note that if maintaining such a history file is your objective, in the Settings > Configure > General Settings dialog you must make sure that Maintain CRD History File is checked.





The **Disable History Feature** button at the top of the dialog shown above is a toggle device. It should be clicked if you prefer not to build the point history file. Clicking it a second time changes it back to saying **Enable History Feature**. You can also choose **Delete History File** to delete the file altogether. By clicking any point from the list, as shown in the Points tab example above, and then selecting **History**, you will be given the history for that specific point. Double-clicking on any command shows the details. Clicking on **Details** also shows the selected command's details. **Undo thru Selected** will undo the effect of all of the commands up through and including the selected command. The changes from the undo command are themselves then added to the command list and can be undone in the future.

Point Protect Toggle: This option, located at the bottom-left of the main *Coordinate File Utilities* dialog, toggles point protection on and off. With this option on, when attempting to store a point with a point identifier (point number) that already exists in the current coordinate file, the following dialog will be displayed.



Overwrite with new coordinates will update the existing point number with the new location of the point.

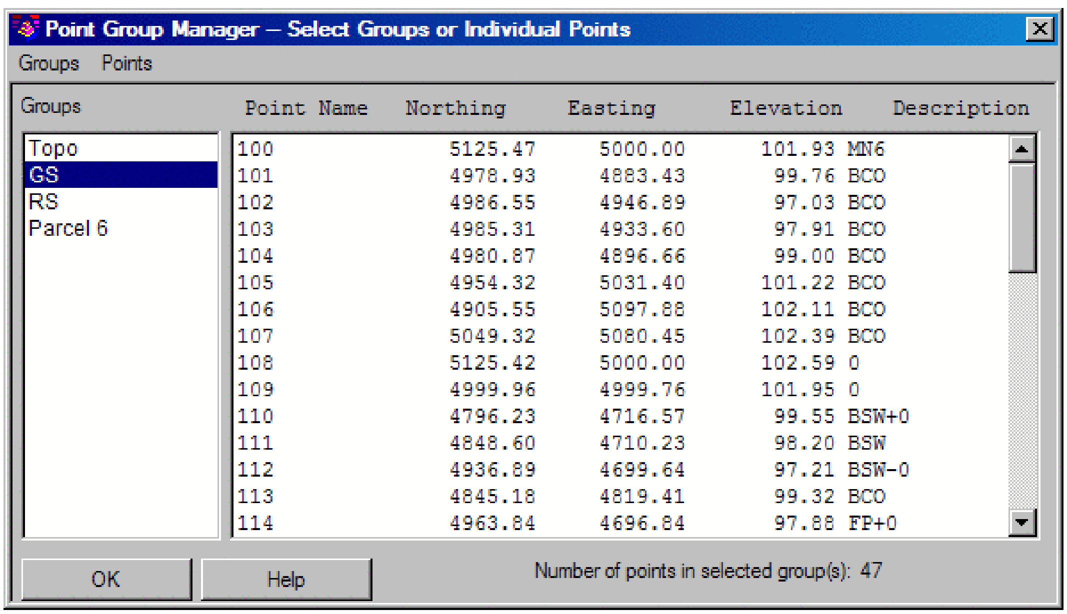
The **Use Another Number** field displays the point number that will be used if the Use Another Number option is selected. This number will depend upon the option chosen from the **Another Number From** settings. If **Next Available** is chosen, the next available number will be displayed in the Use Another Number Field. If there are number gaps in the coordinate file this number will not be the next highest number in the file. For example if points 1-10 and 20-30 exist in the crd file leaving a gap from 11-19, the Next Available number would be 11. If the desired point number, in this example, is 31, then the option of **End of File** would be selected.

The **Overwrite All** and **Renumber All** options apply when more than one point with the same number exists in the coordinate file. These options are helpful when importing points into existing CRD files.

Pulldown Menu Location: Points
Keyboard Command: cfu
Prerequisite: None
File Names: \lsp\crdutil.lsp, \lsp\crdutil.arx, \lsp\scadcfu.dcl, \lsp\scadfile.dcl

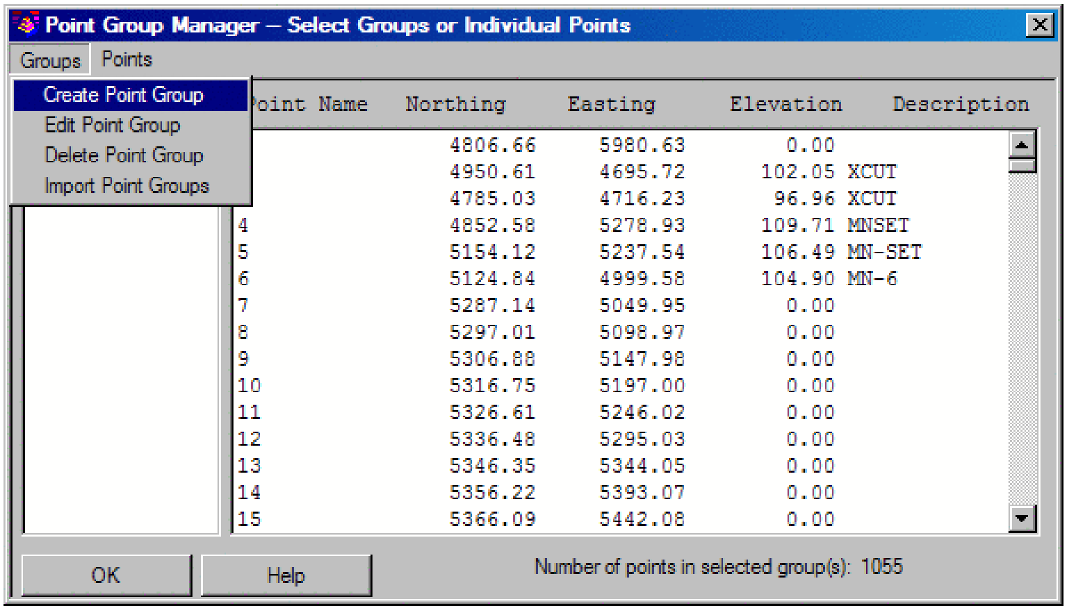
Point Group Manager

This command is used to create point groups based on inclusion and exclusion filters. The manager can perform various functions on these point groups. Also point groups can be referenced by group name in other commands such as Field to Finish and Data Collection.



Groups Pulldown

Create Point Group: This option creates point groups. When selected, the New Point Group dialog box is displayed.



Group Name is the name of Point Group to create.

Description is the description of Point Group to create.

Use the **Include Tab** to define the filters to be applied when creating the point group. Inclusion rules are applied before the exclusion rules.

When **Include All** is toggled on, all points in the coordinate file will be included in the selection.

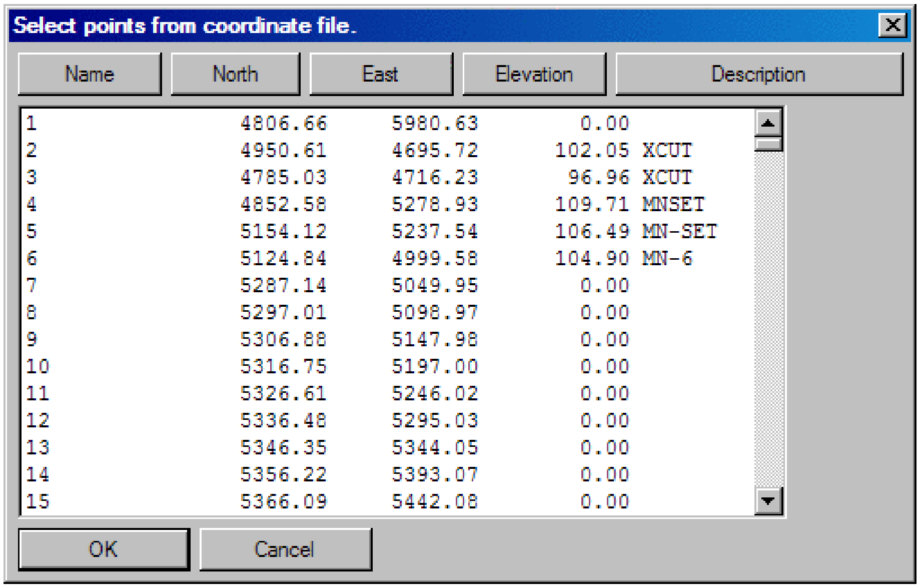
When **Point List** is toggled on, an option of defining the point list must be selected.

DWG: Select allows for manual selection of the points to include from the drawing. The points must be drawn on the screen prior to using this option. All standard AutoCAD selection tools, are available for selection of the points.

DWG: Add Within Circle allows for selection of the points to include by a user defined circle. The circle is defined by specifying the center and radius of the circle. The radius can be defined by entering in a numeric value or by picking on the screen. Points must be drawn to the screen prior to using this option.

DWG: Add Within Polyline allows for the selection of points to include by referencing a closed polyline. All points located within the closed polyline will be included in the selection. Prompts for the inclusion polyline and the exclusion polyline will display. The inclusion polyline limits of the selection area. The exclusion polyline defines the area to exclude within the inclusion polyline. Points must be drawn to the screen prior to using this option.

CRD: Select allows for manual selection of the points to include from a point list. Standard window selection tools are available for selecting the points to include.



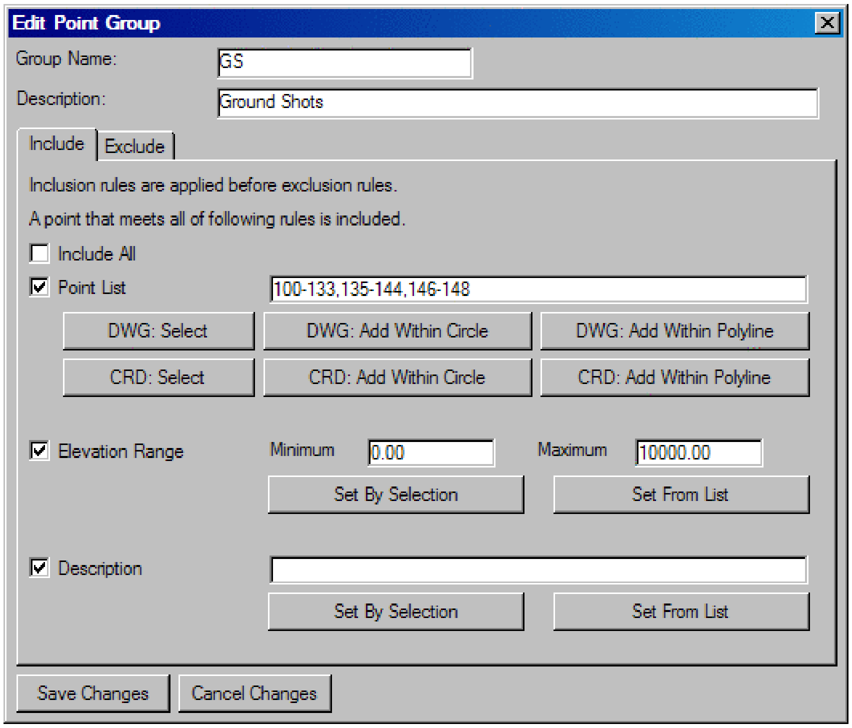
CRD: Add Within Circle allows for selection of the points to include by a user defined circle. The circle is defined by specifying the center and radius of the circle. The radius can be defined by entering in a numeric value or by picking on the screen. The points do NOT have to be drawn to the screen prior to selection.

CRD: Add Within Polyline allows for the selection of points to include by referencing a closed polyline. All points located within the closed polyline will be included in the selection. Prompts for the inclusion polyline and the exclusion polyline will display. The inclusion polyline limits of the selection area. The exclusion polyline defines the area to exclude within the inclusion polyline. The points do NOT have to be drawn to the screen prior to selection.

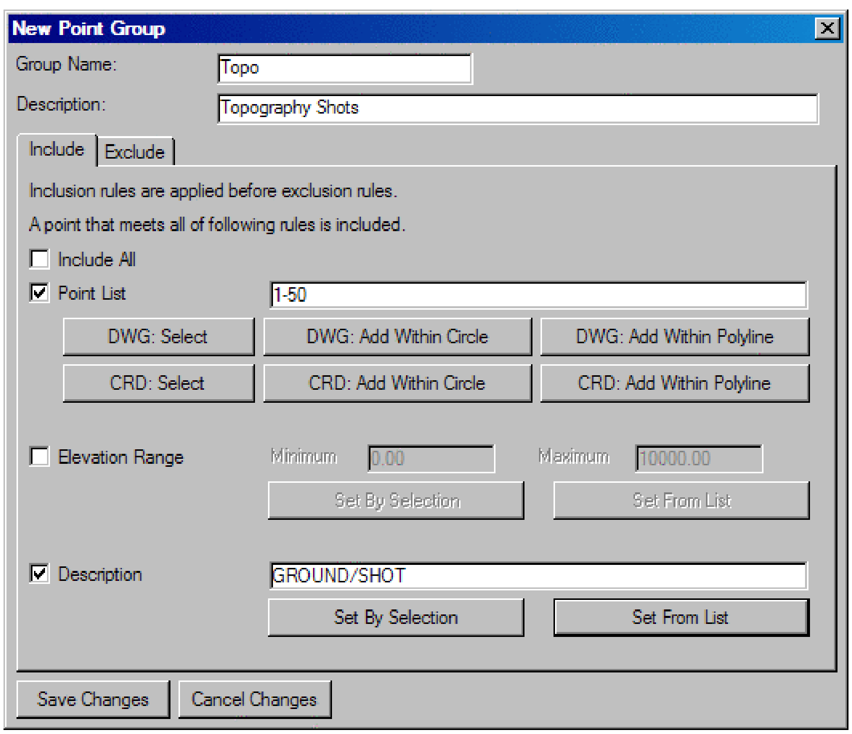
Elevation Range allows for the selection of points within a specified elevation range to be included in the group. The minimum and maximum elevations can be entered manually in their respective data fields. The minimum and maximum values can also be specified by the Set By Selection and Set From List options.

Set By Selection allows for selection of points to include in the group from the drawing. The points must be drawn to the screen prior to using this selection method. Standard AutoCAD selection methods are available.

Set From List allows for selection of points to include in the group from a point list. Standard Windows selection tools are available with this option.



The **Description** option allows for a selection of points to include based upon the description of the point. The description to filter for can be entered in the data field or by using the Set By Selection and/or the Set From List options described above.

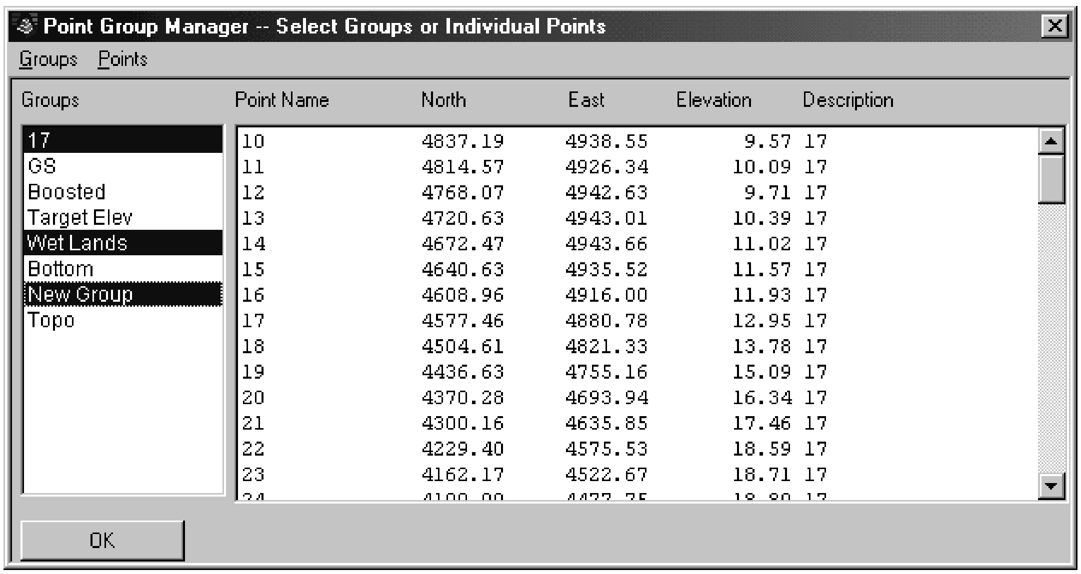


The **Exclude Tab** allows for defining rules that pertain to the points to be excluded from the Inclusion selection. After defining the inclusion rules for the group, the options on the Exclude tab can be used to filter for points to exclude from the group. For example, if the inclusion rules call for all points within the elevation range of 8 to 12, an exclusion rule can be set to exclude the points on elevation 9 or with the description tree. The options on this tab work exactly like the options on the Include tab. Please refer to the Include tab definitions for further instruction.

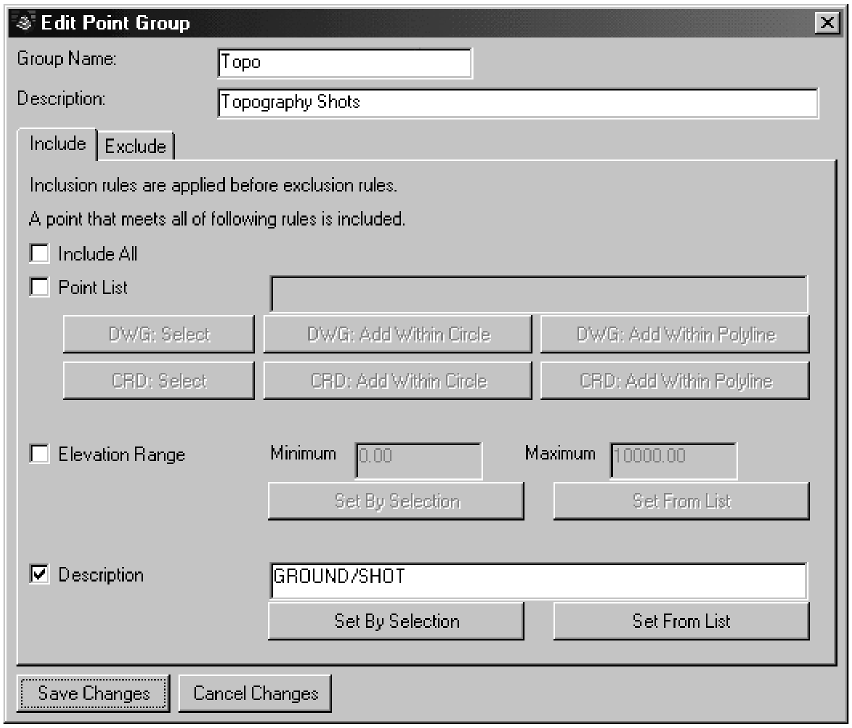
Save Changes saves the point group to the group name specified based upon the Inclusion and Exclusion rules specified.

Cancel Changes discards specified rules and changes and goes back to the Point Group Manager dialog.

Edit Point Group allows for editing of existing point groups. From the list of available groups, highlight the group or groups to edit. When complete with the first group, if more than one is selected, selecting the Save Changes option will save the changes to the active group and switch to the next group in the selection set.



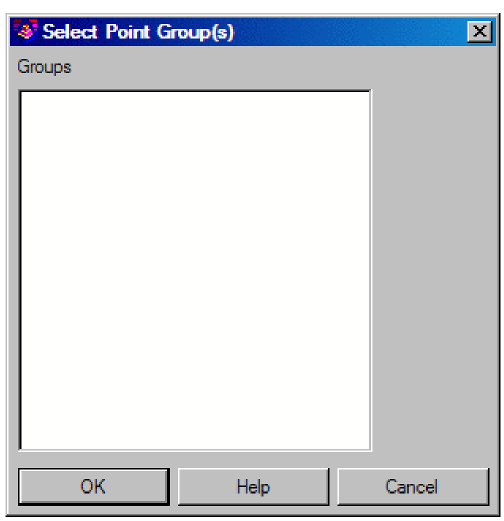
From the Groups pulldown, select Edit Groups, the Edit Group dialog box will now appear.



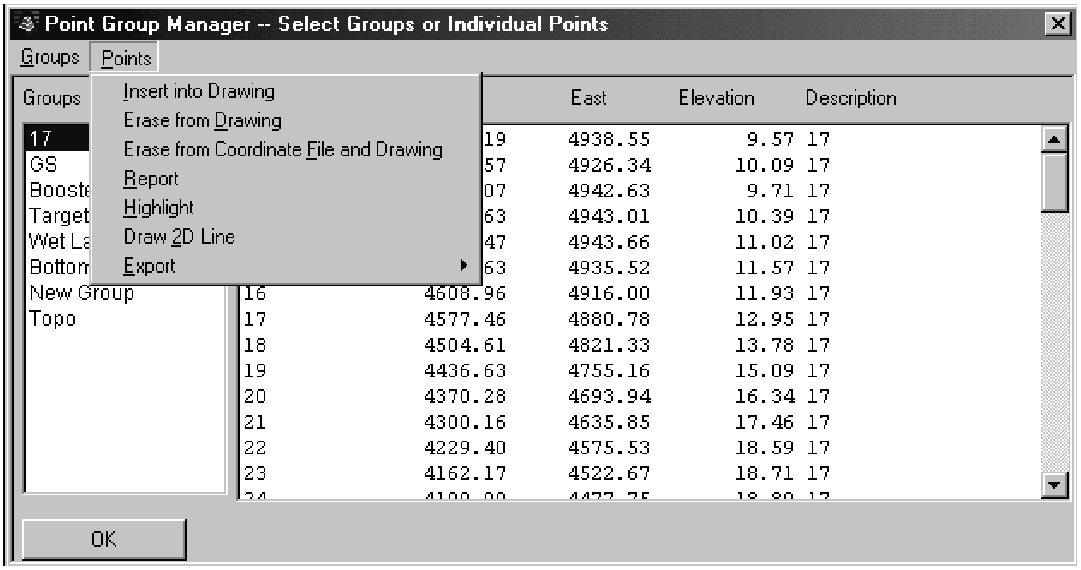
See Create Point Groups for further definitions of the available options.

Delete Point Groups deletes specified groups for the existing group list. One or more groups can be deleted at one time.

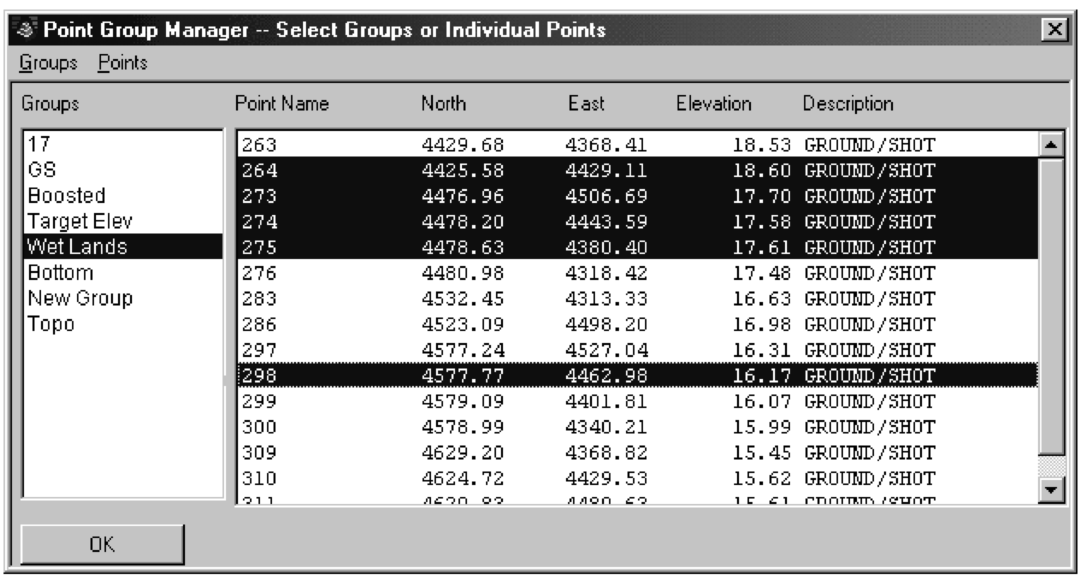
Import Point Groups allows for importing filters from point group manager settings of other coordinate files. This is a useful option when coordinate files are going to contain same point group names with the same filters. This option only brings in the filters into the point group manager, it does not import actual points into the coordinate file by group name. Existing points in the active coordinate file that meet the filter definitions of the imported point groups will automatically be added to the corresponding group.



Points Pulldown



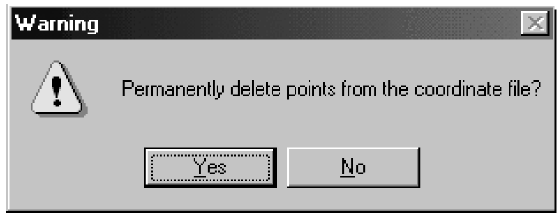
The **Insert into Drawing** option draws the points in the group in the drawing. Individual points or point ranges can be selected from the group to be erased from the drawing. For example points 264-275 and point 298 contained in group Wet Lands are tagged to be erased from the drawing in the following figure.



The symbol to be used and the attribute layout are determined by the Point Default Settings. The symbol size and the point attribute size are determined by the settings in the Drawing Setup routine.

Erase from Drawing erases specified point group/groups or specified points from within the group from the drawing.

Erase from Coordinate File and Drawing erases the points in the specified group/groups or specified points from within the group from the drawing and will also permanently delete the points from the CRD file. You will be prompted with a warning as follows:



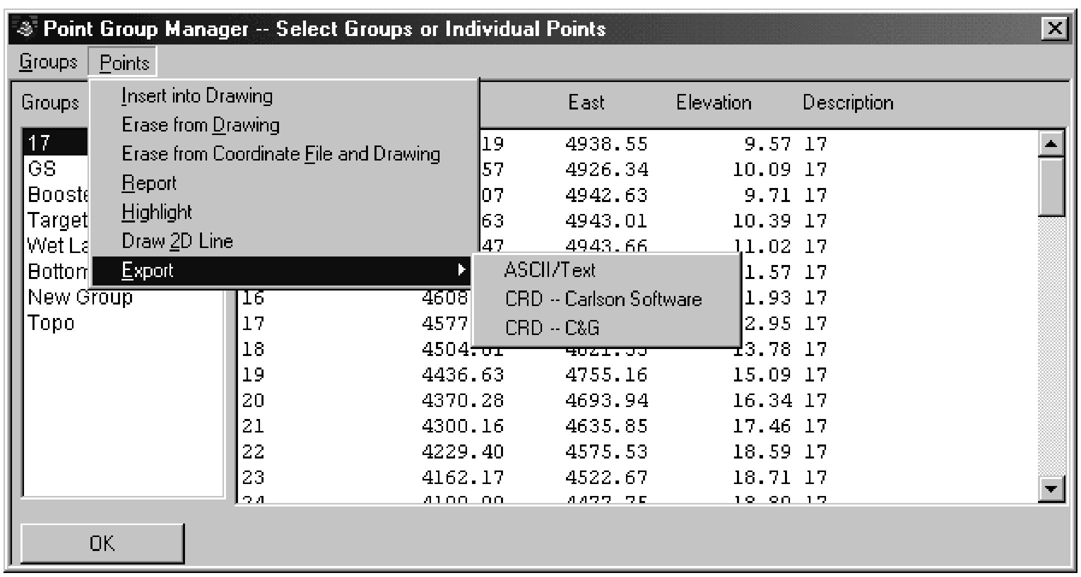
Selecting **Yes** will complete the command and erase the points from the screen and also the coordinate file. Selecting **No** will cancel the command leaving the drawing and the coordinate file unchanged.

The **Report** option will generate a point list of the points contained in the selected group/groups or specified points from within the group.

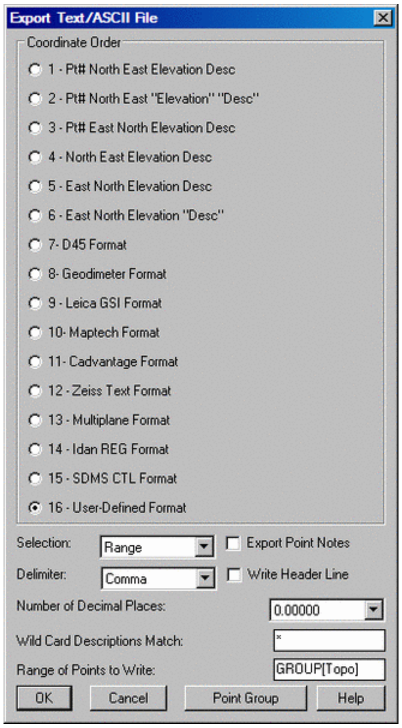
The **Highlight** option highlights the specified objects in the drawing. This makes them distinguishable from the other points on the screen.

The **Draw 2D Line** option draws a 2d polyline between the points contained in the group/groups or between specified points in a group.

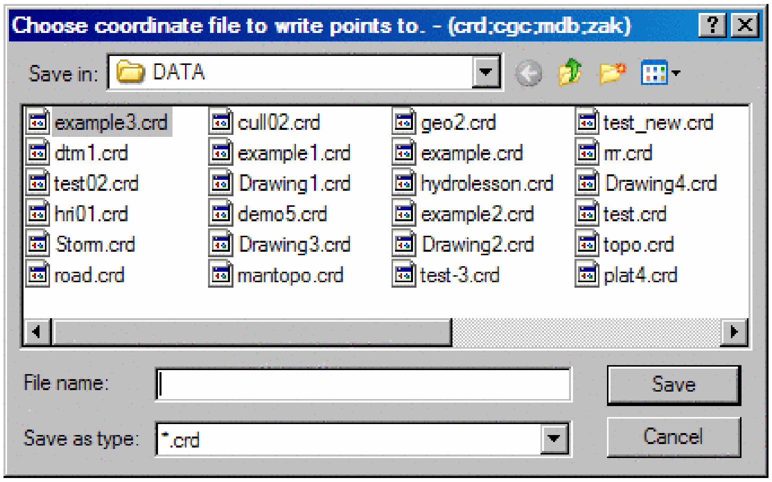
The **Export** command exports the selected group/groups or the specified point(s) or range of points from within the group to various formats. The available formats are ASCII/Text, Carlson Software CRD and C&G CRD files.



When **ASCII/Text** is selected, the Export Text/ASCII File dialog box will be displayed. Please refer to the Export Text/ASCII File section of the manual for more information.



The **CRD-Carlson software** command writes the selected group/groups or the specified point(s) or range of points within the group to a new Carlson formatted CRD file.



Specify the file name of the CRD file to create and press save.

CRD-C&G writes the selected group/groups or the specified point(s) or range of points within the group to a new C&G formatted CRD file.

Specify the file name of the CRD file to create and press save.

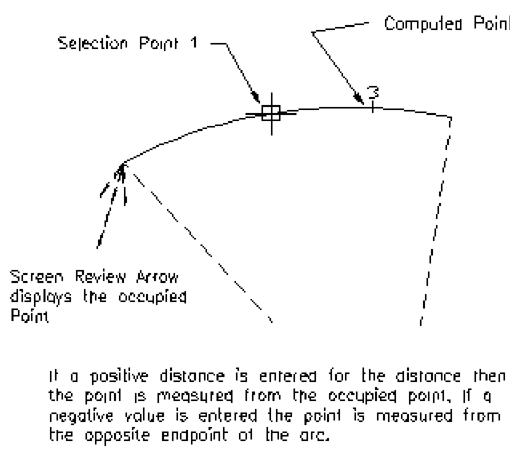
Pulldown Menu Location: Points
Keyboard Commands: pgm
Prerequisite: A coordinate file
File Names: \lsp\crdutil.arx

Point on Arc

This command locates a point on an arc. You can select an arc entity, an arc polyline segment or enter three points to define an arc. After the arc is defined, the screen preview arrow shows the occupied point and the distance to solve for is entered. The command then displays the curve information and locates/inserts a point symbol at the computed point. When prompted for the distance, use a positive value if the distance is from the 1st endpoint (PC the one highlighted by the screen preview arrow) and a negative value if from the 2nd endpoint (PT).

Prompts

Define arc by, Points/<select arc or polyline>: *pick arc or polyline arc segment* Pick a point on the arc somewhere near it's midpoint. The preview arrow points to the 1st endpoint.
Precede distance with minus sign if distance from 2nd endpoint.
Distance along arc from 1st point: *100*
The command then plots a point at the computed distance.



Pulldown Menu Location: COGO > Interpolate Points
Keyboard Command: ptarc
Prerequisite: None
Filename: \lsp\loconarc.lsp

Divide Between Points

This command divides the distance between two points and inserts one of the point symbols at the specified distances. It can also interpolate elevations (to interpolate the elevations, the points picked must be at their real Z axis elevation).

Prompts

Interpolate elevations [Yes/<No>]? *hit Enter*

Point to divide-interpolate from?
Pick point or point number: *1*

```
 PointNo. Northing(Y) Easting(X) Elev(Z) Description
1         4252.76     4158.32    0.00
```

Point to divide-interpolate to?
Pick point or point number: *pick a point*
Number of Segments-Divisions: *3*
Enter Point Description <>**:***hit Enter*
The command then locates two points.

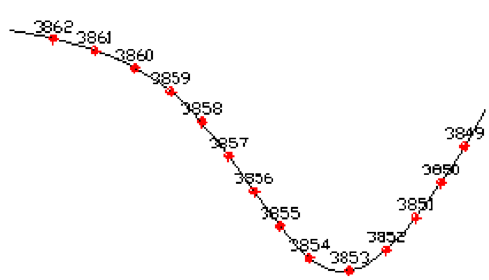
Pulldown Menu Location: COGO > Interpolate Points
Keyboard Command: divlin
Prerequisite: 2 points
File Name: \lsp\divlin.lsp

Divide Along Entity

This command locates points along an entity such as a line, polyline, spline or arc. You must specify the number of divisions.

Prompts

Interpolate Elevations [Yes/<No>]: *press Enter*
Select Entity to Divide: *pick point on entity*
Number of Divisions/Segments: *15*
The command then locates 14 points.

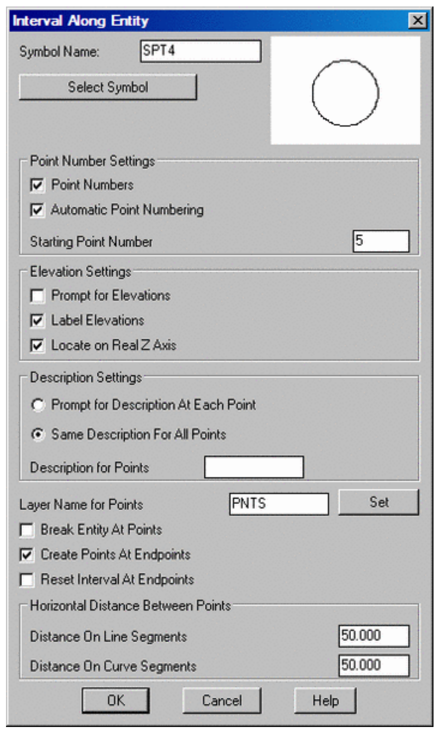


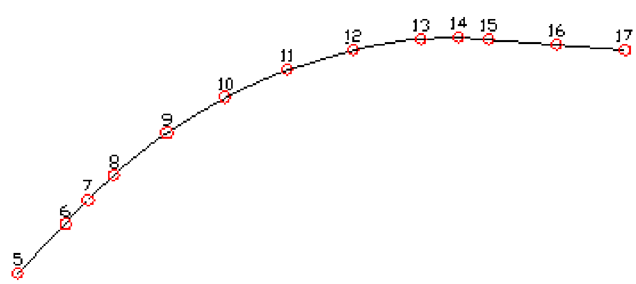
Pulldown Menu Location: COGO > Interpolate Points
Keyboard Command: divent
Prerequisite: 2 points if you want to interpolate elevations
File Name: \lsp\divent.lsp

Interval Along Entity

This command creates points at a specified distance along an entity such as a line, arc, spline or polyline. The points are listed out on the text screen, stored in the current coordinate (.CRD) file and drawn on the screen. For example, you might use this command to locate lot corner points along a frontage line. When Break Entity at Points is checked, the selected entity will be broken at every located point. When Create Point at Endpoint is checked, points will also be located at the endpoints of the selected entity. Horizontal Distance Between Points allow you to specify the distance between located points. There is also an option to create points on curved portions of the centerline at a different interval than on tangent portions (to reduce chord lengths, a shorter interval may be suitable for curves).

For improved descriptions on the points, there is an option, in this main dialog, allowing you to determine whether or not to label elevations on the new points. And for the purposes of describing the points, there is an option that allows you to set the same description to all of the points. For more options related to points, see *Point Defaults* under the Points pulldown.





Create Points at Endpoints turned on

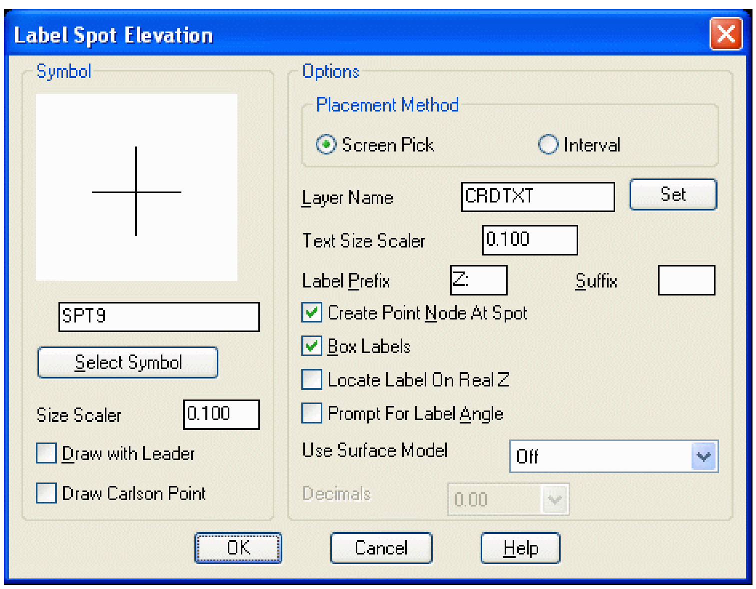
Prompts

Select entity near endpoint which defines first station.
[nea on] Select Entity to Interpolate Points: *select entity*
[nea on] Select Entity to Interpolate Points: Locating 13 Points
The command locates points along the selected entity.

Pulldown Menu Location: COGO > Interpolate Points
Keyboard Command: ptint
Prerequisite: An entity
File Name: \lsp\ptatint.lsp

Spot Elevation

This command allows you to label points with their elevation. The point can either be digitized from a drawing, picked on a screen or specified by a point number. The command first prompts you the **Label Spot Elevation Dialog** for entering layer name, label prefix and suffix and symbol types etc. Click OK to start. After specifying the point, the command prompts you to enter the elevation if its elevation is unknown and then pick an angle from the location of the point to label the elevation. You can repeat labeling points until you press **Enter** to finish.



Prompts

Label Spot Elevation Dialog
Specify a layer name, label prefix and suffix and select the spot symbol.
Point to Label ?
Pick point or point number: *2* Enter a point number.
PointNo. Northing(Y) Easting(X) Elev(Z) Description

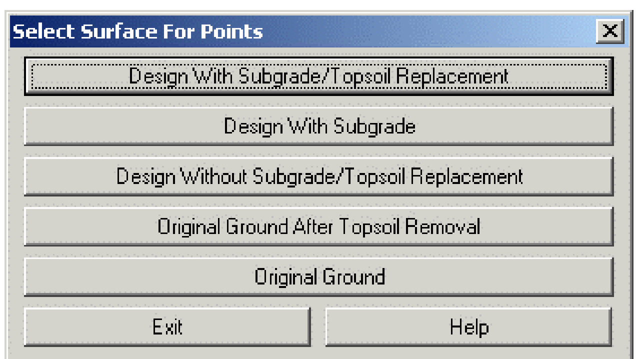
2 1231.16 1099.17 30.00 bb
Note: If the point number you entered is not in the drawing, you will be prompted again to pick point or enter a point number.
Elevation <30.000>: *press enter*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *pick a point on the drawing*
Elevation <0.000>: *enter elevation*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *press enter to finish*

Keyboard Command: LABSPOT
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Create Points On Surface

Function

This command allows you to create points on a selected surface.



Select the surface that you want to create points on, either Design With Subgrade/Topsoil Replacement, Design With Subgrade, Design Without Subgrade/Topsoil Replacement, Original Ground After Topsoil Removal, and Original Ground. This command will calculate the z coordinate of any point that falls within the surface model.

Points can created at various user-specified points or at a specific interval. For individual spot elevations, the user picks or enters the x,y coordinates for each spot elevations. The elevation at the current position of the crosshairs is displayed in real-time in a small window. For interval spot elevations, the alignment for the intervals is defined by a polyline. Then you will be prompted for the interval along the polyline, the number of left offsets and unit interval, and the number of right offsets and unit interval.

Prompts

Command: spotelv

Layer for points <POINTS>:

Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Starting Point Number <1>:

Pick spot elevations or interval along polyline [<Pick>/Interval]? i

Pick the centerline polyline:
Enter interval along polyline <50.0>:

Enter number of left offsets <0>: 4

Enter left offset interval <50.0>: 20

Enter number of right offsets <4>:

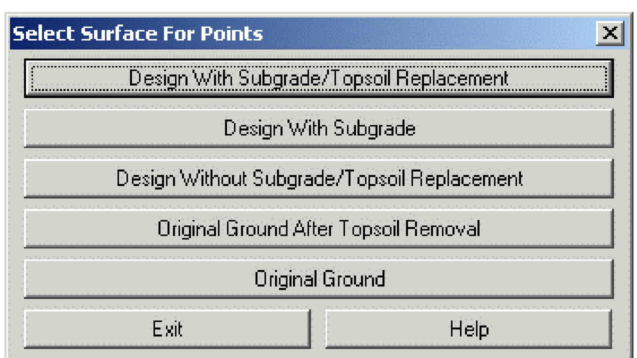
Enter right offset interval <20.0>:

Prerequisite: a surface

Keyboard Command: spotelv

Set Point Elevations To Surface

This command allows you to set point elevations on a selected surface.



Select the surface that you want elevations to be added, either from Design With Subgrade/Topsoil Replacement, Design With Subgrade, Design Without Subgrade/Topsoil Replacement, Original Ground After Topsoil Removal, and Original Ground. Next, pick the points to convert and this command will set the elevations of the points to the surface.

Prompts

Command:
3DCONVERT
Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Select points to convert.
Select objects: Specify opposite corner: 86 found, 25 groups
11 were filtered out.

Select objects:

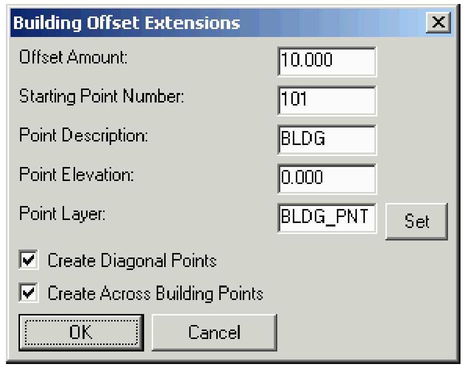
Converting points...
Converted 25 points.

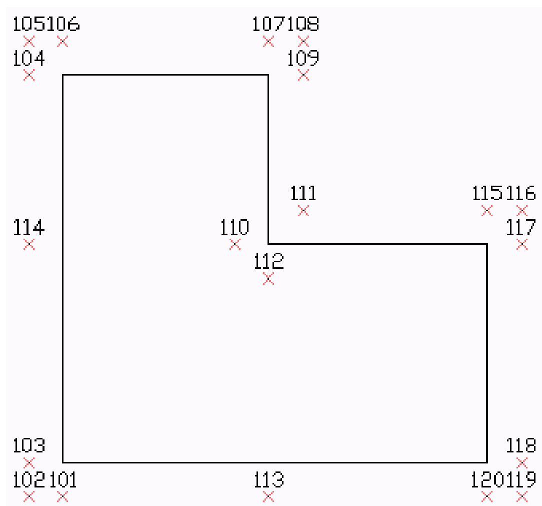
Prerequisite: a surface with an elevation and points to convert

Keyboard Command: 3DCONVERT

Building Offset Extensions

This command is used to calculate building corner offset points that are extensions of the building faces. This command uses building perimeters that are drawn as closed polylines. The point are stored to the current coordinate file and draw on the screen. There is a dialog for setting the parameters. The Offset Amount is the distance that the offsets are extended past the end of the building line. The Starting Point Number is the point number to begin storing from. The Point Description and Elevation are assigned to all the new points and the Point Layer is used for all the drawn points. Offset points are always created as extensions of the building lines at the corners. Offset points can optionally be created at the diagonals of corners and across to the other side of the building for inside corners. In the example show here, points 101, 103, 104, 106, 107, 109, 110, 112, 115, 117, 118 and 120 are corner extension offset points. Points 102, 105, 108, 111, 116 and 119 are diagonal points. Points 113 and 114 are across building points.





Prompts

Building Offset Extensions dialog
Select building perimeter linework.
Select objects: *make selection*

Pulldown Menu Location: COGO
Keyboard Command: bldg_pnts
Prerequisite: A polyline perimeter that represents a building
File Name: \lsp\poly3d.arx

Import Text/ASCII File

This command converts point data from an ASCII text file into the current Carlson coordinate (.CRD) file. Each line of the text file can contain any combination of point number, northing, easting, elevation and description. All point information should be on one line with the values separated by a comma, space or other delimiter. Under the Source File Format setting you can choose from some specific formats or User-Defined. For User-Defined, the format of the text file is specified in the Coordinate Order field where the value identifiers are listed with the appropriate delimiters. For example:

For a text file with northing, easting, elevation and comma delimiters:

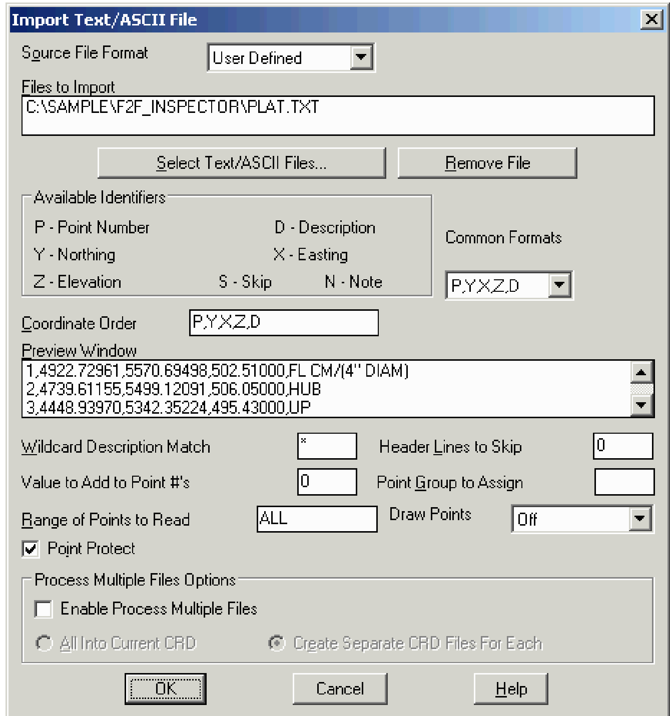
5100.0,5150.5,485.1
5127.1,5190.3,487.3
The Coordinate Order would be:
Y X Z

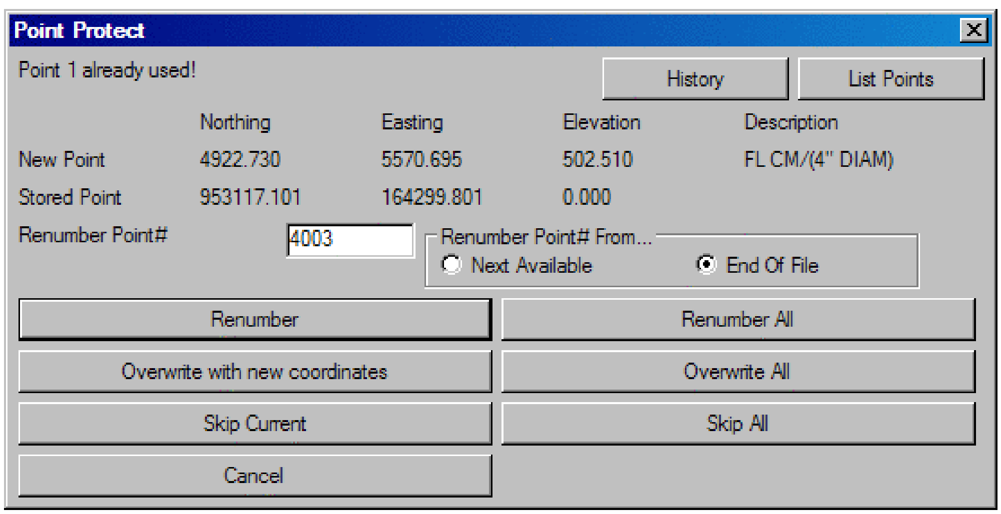
For a text file with point number, easting, northing, elevation, description and space delimiters:
1 5000.0 5000.0 490.3 TRAV
2 5030.4 4930.5 495.5 TRAV
The Coordinate Order would be:
P X Y Z D

Common formats can be selected from the Common Format List. All the lines in the text file should contain only point data and any header lines should be removed. To read the text file, pick the Select Text/ASCII File button and choose the file to read. Then the selected file is displayed in the Preview Window to help with filling out the Coordinate Order. When the Coordinate Order is set, click OK to read the text file. The Wild Card Descriptions Match allows for only point with matching descriptions to be imported. With Point Protect active, the program will check if a point number already exists in the CRD before importing the point. If a point conflict is found, you can either assign a new point number or overwrite the old point. The Value to Add to Point Numbers allows you to renumber the points as they are imported. The Header Lines to Skip value is the number of lines not to be processed at the start of the text file. The Point Group To Assign option will create a point group with the specified name for the coordinate file containing the point numbers imported with Import Text/ASCII File.

Multiple files can be imported at once. To do this toggle on the Enable Process Multiple Files option. After selecting the Text/ASCII Files button, you can select multiple files by using the Shift or Ctrl keys while picking files. You can also run Select Text/ASCII Files multiple times allowing for selection of files located in different locations. The files to import are listed in the top scroll display window. The point data from all the import files can be stored to the current CRD file or to separate files for each import file. The separate file option will name the resulting CRD files with the same name as the import file with a .CRD file extension. For example, the import file job125.txt would create job125.crd. The CRD file will be created in the same location as that of the selected text file to import.

The special formats of Leica .gsi files, TDS .cr5 files, Geodimeter .obs/.raw files, Laser Atlanta .txt files, Trimble .pos files, Zeiss .txt files, Traverse PC .trv files, Maptech, Benchmark .dat files and Cadvantage .cog files can be directly imported by choosing that File Format at the top of the dialog.





Pulldown Menu Location: Points
Keyboard Command: readpt
Prerequisite: A text file to read
File Name: \lsp\crdutil.arx

Import TDS RD5 File

Function

This command converts a TDS RD5 road file into TakeOff centerline (.CL) and profile (.PRO) files.

Prerequisite: a TDS RD5 file

Keyboard Command: importrd5

Import LDD MDB File

Function

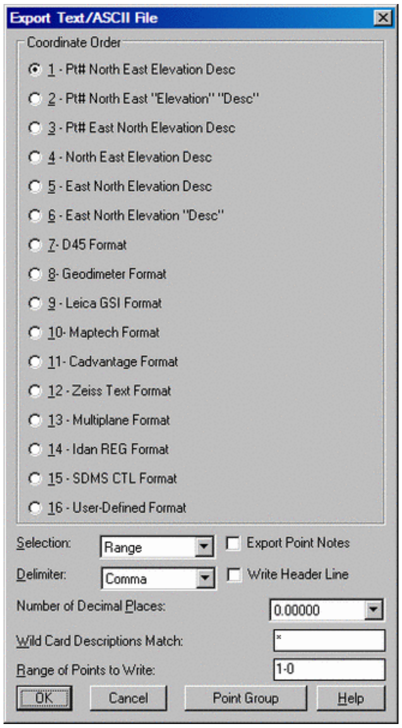
This command converts a LDD MDB file into TakeOff centerline (.CL) and profile (.PRO) files.

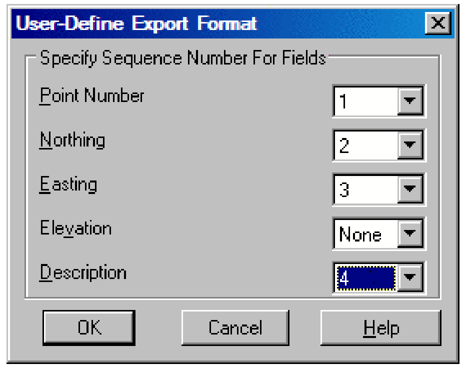
Prerequisite: a LDD MDB file

Keyboard Command: ldd_crd

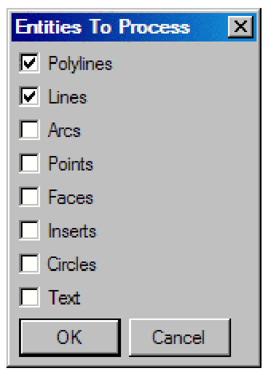
Export Text/ASCII File

This command outputs point data from the current Carlson coordinate file to an ASCII text file. Specify the type of file to write with the Coordinate Order radio buttons. There are several variations on point number, northing, easting, elevation and descriptions as well as specific formats for Leica, Geodimeter, Zeiss, Maptech, D45, Cadvantage, Multiplane and SDMS CTL formats. In addition there is an option, User-Defined Format, to define the order of the fields output. When using the User-Defined format, after selecting OK, the User-Define Export Format dialog will appear.On this dialog, specify the order of the fields by defining a number sequence in each field. You can skip fields and omit data in the output file by leaving None in the sequence field for this data.





Specify the Delimiter of the export file as either Comma or Space in the Delimiter field. There are three Selection Methods provided for the data to export. Specify either Range, Screen Points or Screen Entities in the Selection Field. A Range selection is a user specified range such as 1-10,30-50. A Screen Points selection is made by selecting points from the screen area. The Screen Entities option allows for selection of polylines, lines, arcs, points, faces, inserts and text to export point data from. When the Screen Entities option is selected, the following dialog box will display allowing for the specification of the type of entity to export data from.



A description filter is also available for exporting only points from the range or selection set with certain descriptions. After selecting the OK button, another dialog appears that allows you to specify a new text?ASCII file or to append data into an existing file. The standard file selection dialog allows you to specify the export file name.

Pulldown Menu Location: Points
Keyboard Command: writept
Prerequisite: A Coordinate File (.CRD)
File Name: \lsp\crdutil.arx

Convert AECC POINTs to Carlson Points

Function

This command converts AECC_POINTs to the Carlson format used by Autodesk Land Development Desktop (LDD). Requires successful installation of the proper Autodesk Object Enabler. This component can be downloaded from the Autodesk web site at http://pointa.autodesk.com/

Prompts

Loading AecBase...

Loading AecCivilBase...

1 Convert all or selected points [All/<Selected>]? **A**

Prerequisite: Points on the screen

Keyboard Command: AEC_ PT

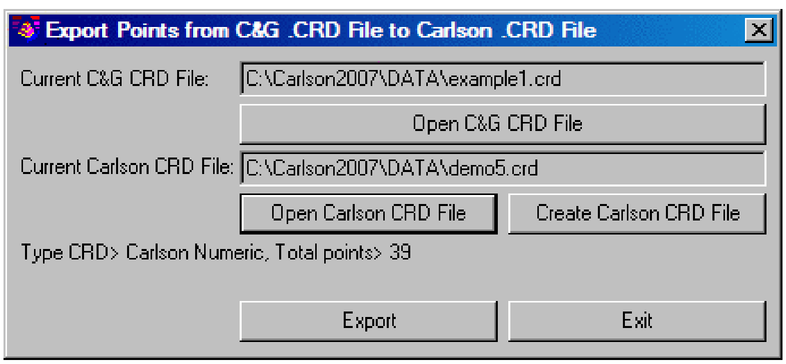
Convert Softdesk to Carlson Points

This command converts Softdesk point blocks in the drawing to Carlson point blocks. These point block formats are similar and converting only requires reordering and renaming the attributes. Softdesk points can also be read into the current CRD file by using the command *Update CRD File from Drawing* in *Coordinate File Utilities*, this updates the CRD file without modifying the screen entities.

Pulldown Menu Location: Points > Convert Point Format
Keyboard Command: 2surv
Prerequisite: Softdesk points
File Name: \lsp\2surv.lsp

Convert C&G to Carlson Points

This command converts C&G Points into a Carlson CRD file.



Specify the existing C&G File to convert by selecting the Open C&G CRD File button. Specify the existing Carlson CRD file to write to, or the new Carlson CRD file to create, by selecting either Open Carlson CRD file or Create Carlson CRD file. Press OK and the conversion is completed.

Pulldown Menu Location: Points > Convert Point Format
Keyboard Command: cg2crd
Prerequisite: A C&G point file

Convert Leica to Carlson Points

This command converts LisCad or Leica point blocks in the drawing to Carlson point blocks. These point block formats are similar and converting only requires reordering and renaming the attributes. Leica points can also be read into the current CRD file by using the command *Update CRD File from Drawing* in *Coordinate File Utilities.* This updates the CRD file without modifying the screen entities.

Pulldown Menu Location: Points > Convert Point Format
Keyboard Command: 2surv3
Prerequisite: Leica points

Convert Eagle Point to Carlson Points

This command converts Eagle Point point blocks in the drawing to Carlson point blocks. These point block formats are similar, and converting only requires reordering and renaming the attributes. Eagle Point points can also be read into the current CRD file by using the command *Update CRD File from Drawing,* found in *Coordinate File Utilities.* This updates the CRD file without modifying the screen entities.

Pulldown Menu Location: Points > Convert Point Format
Keyboard Command: 2surv2
Prerequisite: Eagle Point points
File Name: \lsp\2surv2.lsp

Inquiry Menu 14

List

This command lists the object type, object layer, and*X*,*Y*,*Z* position relative to the current user coordinate system (UCS) and whether the object is in model space or paper space.

The List command reports color, linetype, and lineweight information if these items are not set to BYLAYER. The thickness of an object is displayed if it is nonzero. *Z* coordinate information defines the elevation. If the extrusion direction of the entry differs from the *Z* axis (0,0,1) of the current UCS, the List command also reports the extrusion direction in UCS coordinates. The List reports additional information related to the specific object selected.

Prompts

Command:
LIST
Select objects: 3 found, 1 group

Select objects:

BLOCK REFERENCE Layer: "PNTS"
Space: Model space
Handle = 1F3D
Group = ∗A1
"SPT4"
at point, X=6135023.7190 Y=2190074.2098 Z= 800.0000
X scale factor 5.0000
Y scale factor 5.0000
rotation angle 0d0'0"
Z scale factor 5.0000

BLOCK REFERENCE Layer: "PNTS"
Space: Model space
Handle = 1F4D
Group = ∗A1
"SRVPNO1"
at point, X=6135023.7190 Y=2190074.2098 Z= 800.0000
X scale factor 5.0000
Y scale factor 5.0000
rotation angle 0d0'0"
Z scale factor 5.0000

ATTRIBUTE Layer: "PNTNO"
Space: Model space
Handle = 1F4E
Style = "PTXT"
Font file = TXT
center point, X=6135023.7190 Y=2190077.9598 Z= 800.0000
height 5.0000
value 1
tag PT#

rotation angle 0d0'0"
width scale factor 1.0000
obliquing angle 0d0'0"
flags normal
generation normal

ATTRIBUTE Layer: "PNTELEV"
Space: Model space
Handle = 1F4F
Style = "PTXT"
Font file = TXT
start point, X=6135031.2190 Y=2190071.7098 Z= 800.0000
height 5.0000
value 800

Prerequisite: an entity

Keyboard Command: LIST

Point ID

This command reports complete information pertaining to a Carlson point. Although similar in function to the AutoCAD *ID* command, this routine is much more detailed. With this command, you are given the point number, as well as the northing, easting and elevation coordinates. You also are given the point description, and you are shown the name and the location of the coordinate file for the point.

Prompts

Pick point or point number: *255*

```
 PointNo.  Northing(Y)  Easting(X) Elev(Z)   Description
255       4379.83      4265.48    19.01     GROUND/SHOT
N: 4379.83  E: 4265.48  Z: 19.01
PT#: 255  CRD File: c:\Carlson2008\data\mantopo.crd
```

Pulldown Menu Location: Inquiry
Keyboard Command: PT_ID
Prerequisite: None
File Name: \lsp\pt_id.lsp

Layer ID

This command reports the layer name of the selected entity.

Prompts

Pick entity to read layer: *pick an entity*
Layer: FINAL

Pick entity to read layer: *press Enter to end*

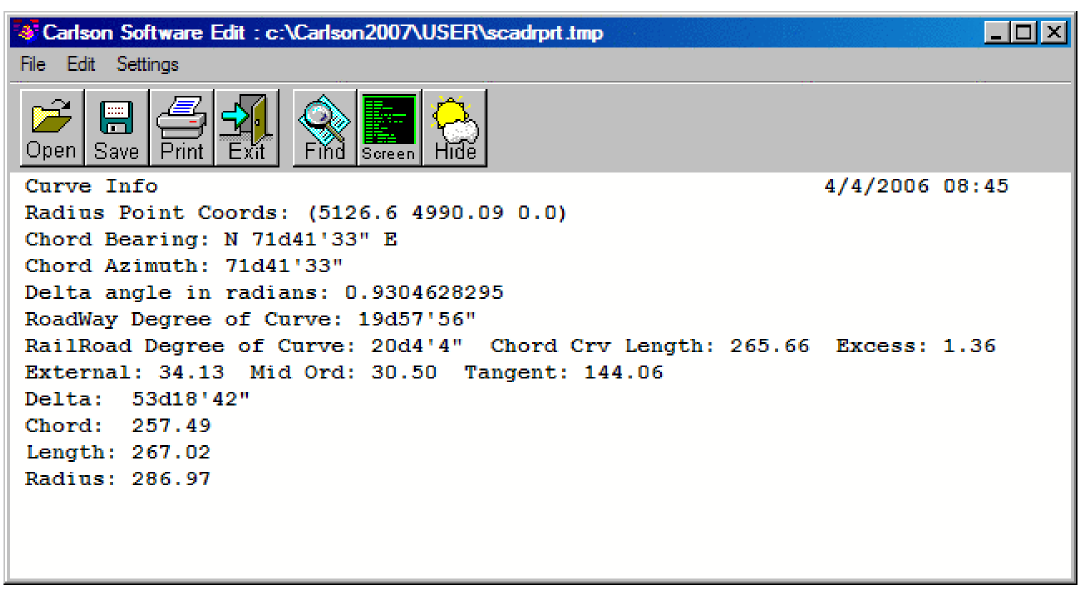
Pulldown Menu Location: Inquiry
Keyboard Command: layerid
Prerequisite: None
File Name: \lsp\surv1.lsp

Curve Info

This command displays information about a curve/arc. The curve can be defined by an arc entity or polyline arc segment or by selecting three points on the arc. The three points can be defined by point number or picked on the screen. The curve data is displayed in the text window with an option to be displayed in the Standard Report Viewer. Click Exit to return to the graphics window.

Prompts

Define arc by, Points/<select arc or polyline>: *select the arc entities*
Endpoint: (4923.81 5193.15 0.0)
Other Endpoint: (5168.27 5274.03 0.0)
Radius Point Coords: (5126.6 4990.09 0.0)
Chord Bearing: N 71d41'33'' E
Chord Azimuth: 71d41'33''
Delta angle in radians: 0.9304628295
RoadWay Degree of Curve: 19d57'56''
RailRoad Degree of Curve: 20d4'4'' Chord Crv Length: 265.66 Excess: 1.36
External: 34.13 Mid Ord: 30.50 Tangent: 144.06
Delta: 53d18'42''
Chord: 257.49
Length: 267.02
Radius: 286.97
Display curve data in report viewer [Yes/<No>]? *Y*



Pulldown Menu Location: Inquiry
Prerequisite: None
Keyboard Command: cinfo
File Name: \lsp\curvinfo.lsp

Polyline Info

This command reports the length and elevation of the selected polyline or line.

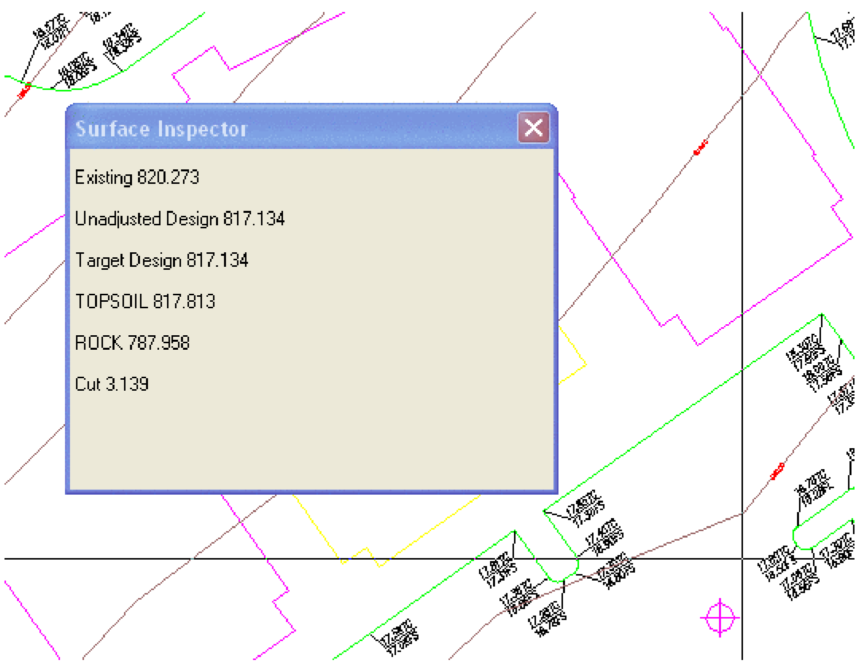
Prompts

Pick Polyline or Line: *pick a polyline or line*
Polyline length: 7702.75 Slope distance: 7702.75 Avg elev: 1700.00 Avg slope: 0.00%

Pulldown Menu Location: Inquiry
Keyboard Command: polylen
Prerequisite: None
File Name: \lsp\polylen.lsp

Surface Inspector

This command allows you to report and optionally label elevations from your drawing. You can analyze all of your different surface files at one time. After running the command, Surface Inspector will begin showing you real-time elevations for each surface as you move the cursor on the screen. If you pick apoint or enter coordinates, the elevation will be labeled along with the surface name.



Surface inspector shows you real-time elevations as you move the cursor over your surface.

Prerequisite: Surface Model (s)
Keyboard Command: surfvals

Surface Report

This command reports a variety of information on each of your different surfaces. This is useful for checking for bad data and the file names of your surfaces. An example is below.

Surface Report 3/10/2005 15:34

Max Cut: 18.327 at 409269.984,207196.674
Max Fill: 1.943 at 409389.586,207248.866

Original Ground After Topsoil Removal
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-ex.flt
Date Modified: Thu Feb 10 10:02:05 2005
File Size: 64,028
Points: 259, Edges: 744, Triangles: 486
Min Z: 184.000 at 409299.790,206879.287
Max Z: 210.000 at 409571.562,207177.240

Design With Subgrade and Topsoil Replacement
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-fn.flt
Date Modified: Thu Feb 10 10:02:08 2005
File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

Original Ground Before Topsoil Removal
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-og.flt
Date Modified: Thu Feb 10 10:02:05 2005
File Size: 64,028
Points: 259, Edges: 744, Triangles: 486
Min Z: 184.000 at 409299.790,206879.287
Max Z: 210.000 at 409571.562,207177.240

Design Without Subgrade or Topsoil Replacement
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-bs.flt
Date Modified: Thu Feb 10 10:02:08 2005
File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

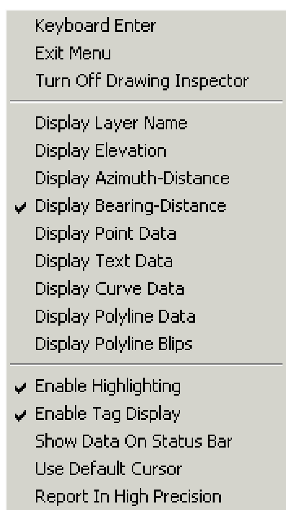
Design With Subgrade
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-zn.flt

Date Modified: Thu Feb 10 10:02:08 2005
File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

Prerequisites: A Surface
Keyboard Command: SURF_STATS

Drawing Inspector

This command reports object properties to you as you move the cursor over an entity. You can simply move the pointer over an entity and the selected property will be displayed either in a pop-up window next to the pointer and/or on the status bar, depending on the selected option. Drawing Inspector is a transparent command that can run while other commands are running. Once Drawing Inspector is started, it will stay active even while running other commands until you turn it off. To turn off Drawing Inspector, run the command again to toggle it off by pick Drawing Inspector from the Inquiry pull-down menu or from the toolbar or by typing the command name, or right-click and choose Turn off Drawing Inspector. The options for this command are set in the menu that pops up by clicking the right mouse button. The available properties are: Layer Name, Elevation, Azimuth-Distance, Bearing-Distance, Point Data, Text Data, Curve Data, 3D Face Data, Polyline Data and Polyline Blips.



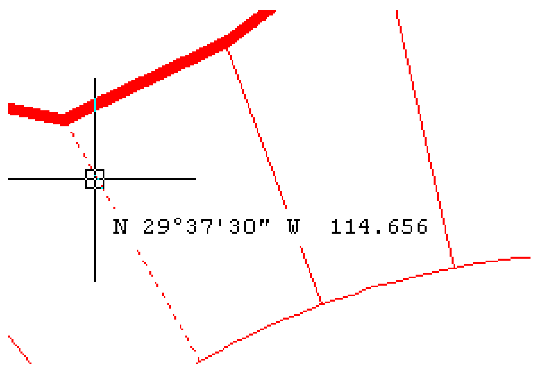
In the *Drawing Inspector* menu, you can choose one or more properties to display.

Display Layer Name: Allows you to display the layer name of the entity.
Display Elevation: Allows you to display the elevation of the entity.
Display Azimuth-Distance: Allows you to display the azimuth and distance of a line.
Display Bearing-Distance: Allows you to display the bearing and distance of a line.
Display Point Data: Allows you to display the coordinate data of point.
Display Text Data: Allows you to display the attributes of text.
Display Curve Data: Allows you to display the radius, arc length, chord length and delta angle of a curve.
Display 3D Face Data: Allows you to display the Z elevations at the face corners.
Display Polyline Data: Allows you to display the end point elevations, horizontal distance, slope distance and slope

ratios.
Display Polyline Blips: Allows you to displays temporary blip plus marks at the vertice locations of polylines.

In the *Drawing Inspector* menu, you can also choose how the property information is reported.

Enable Highlighting: Allows you to highlight the object that the *Drawing Inspector* is reporting.
Enable Tag Display: Enables you to view the information next to the cursor on the screen.
Show Data On Status Bar: Enables you to view the information on the status bar, in the lower corner of the screen.
Use Default Cursor: When enabled, only the AutoCAD cursor shows. When disabled, the mouse pointer is also shown.
Report In High Precision: When enabled, displays 8 decimals on distance and 4 decimal seconds on angles.



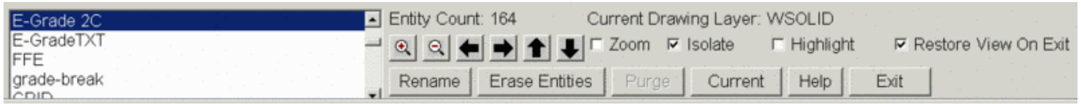
Example of Drawing Inspector reporting
Bearing-Distance using the Tag Display

Pulldown Menu Location: Inquiry
Keyboard Command: inspector
Prerequisite: None
File Name: \lsp\autotag.arx

Layer Inspector

This command is used to inspect and work with layers in the drawing. This command is ideal when you are working on a very dense and complex drawing which has many layers and you want to review the entities on different layers. In some cases, there will be layers that you would want to erase. Another scenario might be that you'd like to highlight a layer that is hard to find and see.

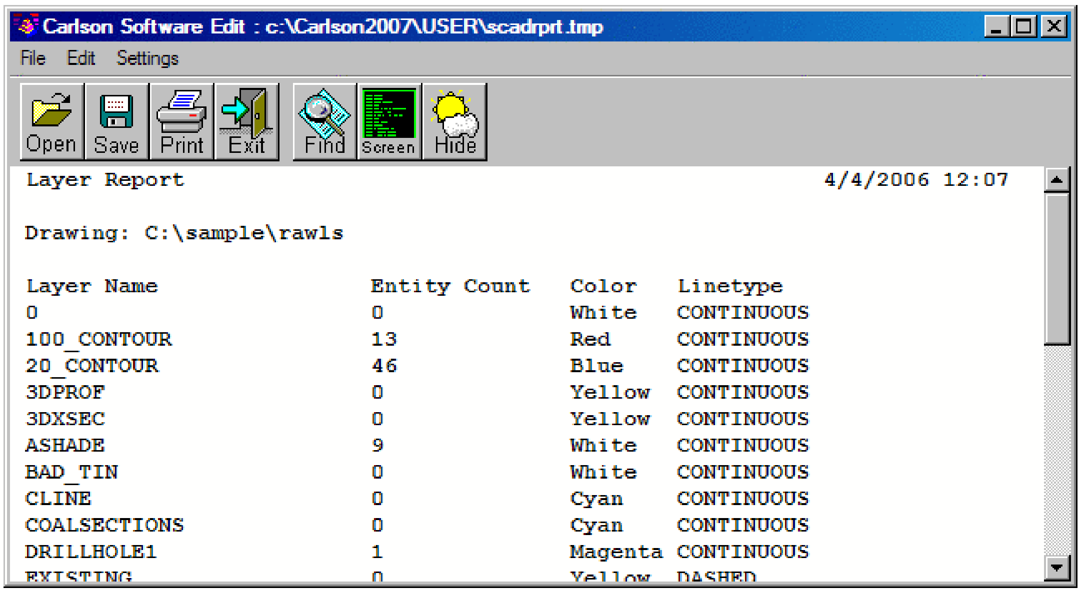
The Layer Inspector command has a dialog that docks to the bottom of the drawing window which keeps the drawing window visible while running the command. On the left of the dialog is a list of all the layers in the drawing. To inspect a layer, highlight the layer name from this list. You can inspect multiple layers at a time by selecting multiple layers in the list using the Shift and Ctrl keys while picking in the list. When a layer is selected, the Entity Count reports how many entities in the drawing are set to that layer. The Zoom toggle will zoom the drawing window to the extents of the entities on the layer. The Isolate toggle will freeze all other layers. The Highlight toggle will highlight all the entities on the layer. The Restore View On Exit will set the drawing window to the original position when Layer Inspector was started. The magnify and arrow buttons are used to zoom in/out and pan the drawing window. The Rename button allows you to rename the layer. The Erase Entities button will erase all the entities on the layer. The Purge button will purge the layer from the drawing which is only available when there are no entities on the layer. The Current button sets the layer as the current layer for the drawing.



Pulldown Menu Location: Inquiry
Keyboard Command: layer_inspect
Prerequisite: None
File Name: \lsp\contour4.arx

Layer Report

This command generates a report containing each layer name, the number of entities on each layer, the color and the linetype applied to each layer.



Pulldown Menu Location: Inquiry
Keyboard Command: reportlayer
Prerequisite: None
File Name: \lsp\surv1.lsp

List Elevation

Function

This command displays the elevation of a polyline or line. With a 3D polyline, the elevation ofthe 3D polyline at the pick point is reported along with the elevation of each vertex. See also,the *Drawing Inspector* command on the ∗Inq-Set∗ menu.

Prerequisite: an entity

Keyboard Command: LSTELEV

Distance Report

Function

This command reports the horizontial distance, slope distance, and elevation difference between two points.

Prompts

Pick point or enter point number:
(409375.0 207039.0 0.0)
Pick second point or enter point number:
(409400.0 207082.0 0.0)
Horiz Dist: 49.73 Slope Dist: 49.73 Elv Diff: 0.00

Prerequisite: none

Keyboard Command: distrprt

Bearing & Distance

This command reports the slope distance, slope ratio, bearing, azimuth and vertical angle between two 3D points. Pick or enter the coordinates of two points or select a line or polyline segment to calculate between the segment endpoints.

Prompts

Specify bearing-distance from (Line/PLine/<Points>)? *press Enter*
Pick point or enter point number: *pick a point*
Pick second point or enter point number: *pick a point*
Horiz Dist: 233.4 Slope Dist: 233.4 Elev Diff: 0.0 Vert Ang: 0d0'0''
Slope: 0.0% 0.0:1 Bearing: S 71d15'37'' W Azimuth: 198d44'23''

Pulldown Menu Location: Inquiry
Keyboard Command: 3DIST
Prerequisite: None
File Name: \lsp\3dist.lsp

Label Angle

Function

This command will label and report the interior and exterior angles between two directions. The angles can be defined by three points or by two line or polyline segments that have a common endpoint.

Prompts

Define angle by, Points/<select line or polyline>: *pick a polyline segment*

Select adjoining line or polyline: *pick another polyline segment*

Interior: 72d39'46'' Exterior: 287d20'14''

Angle to label (<Interior>/Exterior/None)? *press Enter*

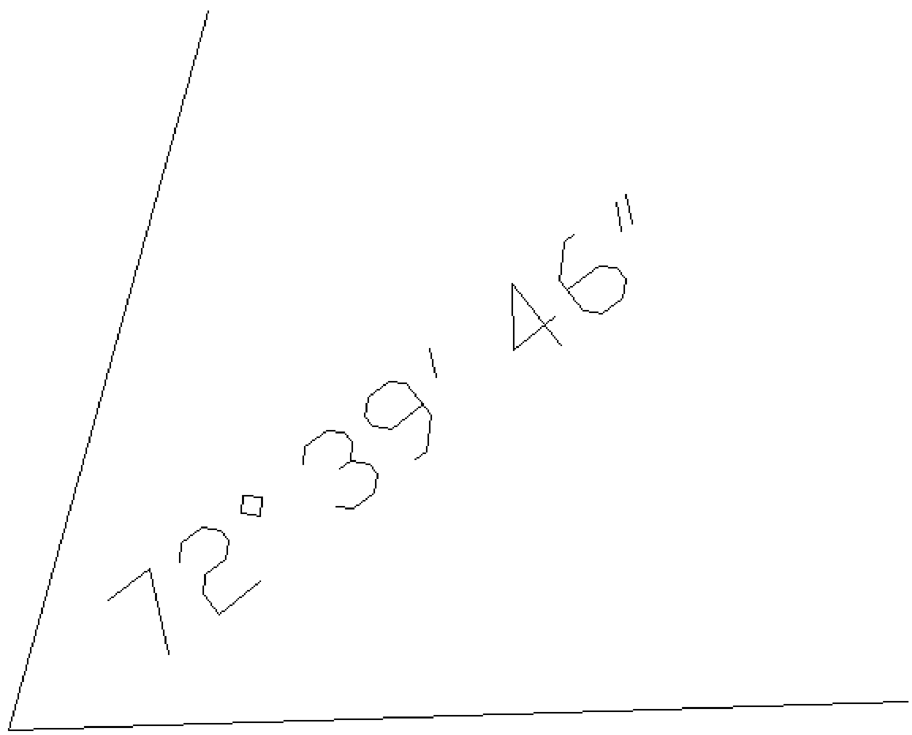
Define angle by, Points/<select line or polyline>: *press Enter to end*

Pull-Down Menu Location: Annotate

Prerequisite: None

Keyboard Command: labang

File Name: \lsp\angle.lsp



Polyline Report

Function

This command generates a report of bearing-distance and curve data for all the point along the selected polyline. The closure is reported between the starting and ending points of the polyline. The polyline area is also reported. After executing the command, by pressing O for options, report deflection angles can be turned on by selecting Y for yes at the prompt. The deflection angle will be displayed in the report along with the bearing and distance of the polyline segments.

Prompts

Starting station <0.0>: *Press Enter*

Options/Select polyline to report: *pick a polyline*

Standard Report Viewer Displays the report for the selected polyline.

Options/Select polyline to report (Enter to End): *Press Enter*

Keyboard Command: plreport

Pull-Down Menu Location: Tools, Polyline Tools

<>**Prerequisite:** a polyline

Polyline Report 05/22/2002 13:24 </>

NORTHING EASTING STATION BEARING DISTANCE

5790.690 4088.812 0.000

N 49°52'11'' E 455.737

6084.426 4437.259 455.737

RADIUS: 367.183 LENGTH: 725.711 CHORD: 613.228 DELTA: 113°14'28''

CHORD BRG: S 73°30'35'' E PC-R: S 40°07'49'' E PT-R: S 73°06'39'' W

RADIUS POINT: 5803.685,4673.919

5910.360 5025.265 1181.448

S 16°53'21'' E 851.945

5095.161 5272.774 2033.393

S 52°00'05'' W 548.996

4757.175 4840.152 2582.389

N 54°19'05'' W 857.964

5257.614 4143.257 3440.353

RADIUS: 695.425 LENGTH: 430.350 CHORD: 423.516 DELTA: 35°27'23''

CHORD BRG: N 72°02'46'' W PC-R: S 35°40'55'' W PT-R: S 00°13'33'' W

RADIUS POINT: 4692.743,3737.624

5388.163 3740.364 3870.702

N 40°52'52'' E 532.394

5790.690 4088.812 4403.097

Closure Error Distance> **0.000**

Total Distance> **4403.097**

Polyline Area: 1201606.5 sq ft, 27.59 acres

Story Stake From Surface Entities

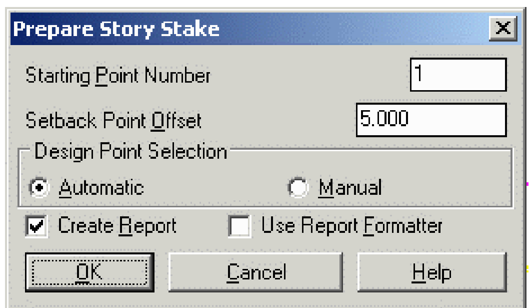
Function

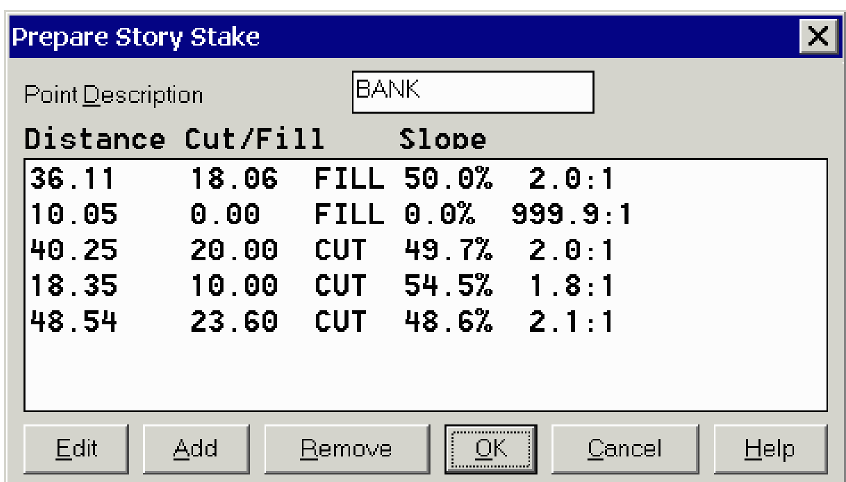
This command creates points with cut/fill information stored in the note fields for the points. Beginning at a point and facing a specified direction, the cut/fill information describes a design surface that is defined by contours and

3D polylines in the drawing.

First, there is a dialog to set the starting point number, setback offset, selection mode and report options. The Automatic selection mode will create a story point for each crossing linework. The Manual mode will show the crossings and let you pick which ones to report.

The program prompts you to pick the starting point followed by a direction point. Then the intersections for all the contours and 3D polylines between these two pointsare calculated and the resulting horizontal distances and slopes are shown in a dialog. In this dialog, you can edit, add or remove these slopes descriptions. The Point Description can also be specified. When OK is clicked, a point in the coordinate file is created at the starting point with this information stored in the note file. An offset point is also created at the specified offset distance back from the starting point. At the end of Story Stake, a report of all the created points and the corresponding cut/fill data is shown if the Create Report option was set. Story Stake does not draw the points in the drawing. These points can be drawn using the Draw-Locate Points command.



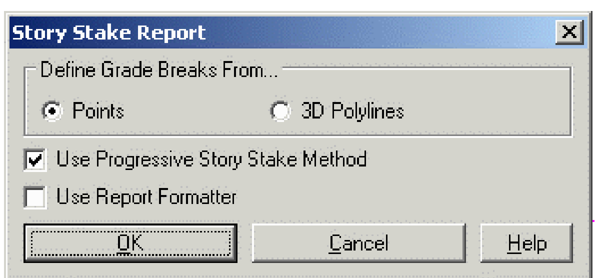


Story Stake By Points/Polyline

Function

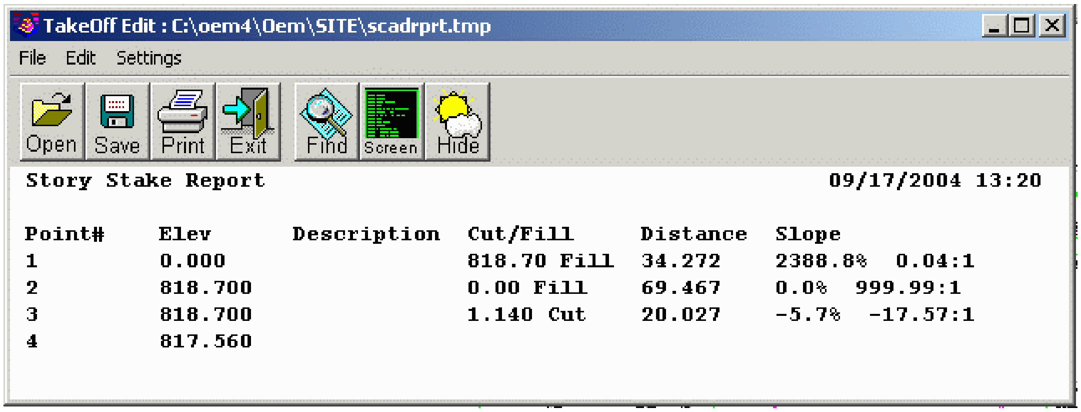
This command creates a story stake report of distances, slope and cut/fill between selected points or along a selected 3D polyline.

First, there is an options dialog. The Progressive method reports each distance from the previous point to the next. The non-progressive method reports the distances from the starting point to each point. The report formatter option lets you customize the fields for the report and output the report to Excel.



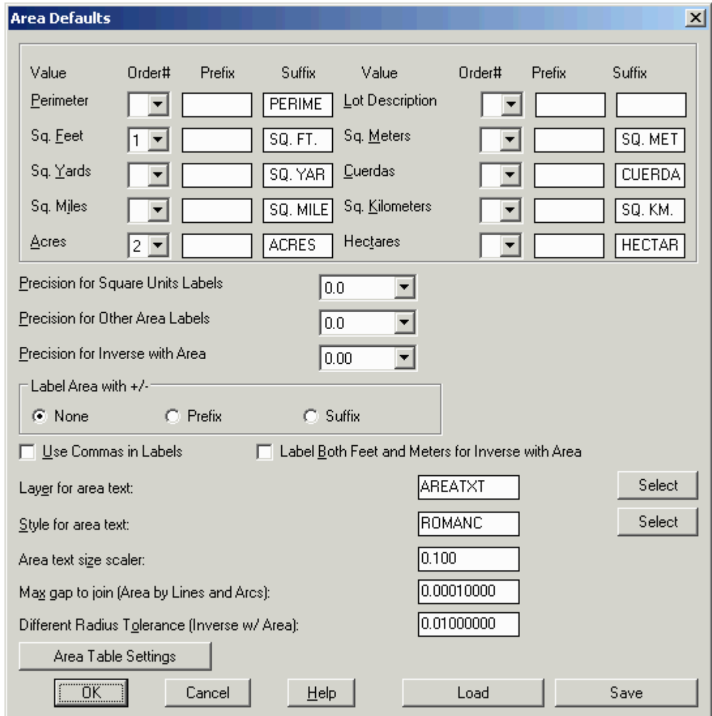
For the points method, there are prompts at the command line to pick the points. The points should be picked one at a time in series from start to end. The points can be entered by point number from the current coordinate file or by screen pick.

For the polyline method, there is a prompt to select the 3D polyline for the grade. The polyline should be drawn in the order of the story stake.

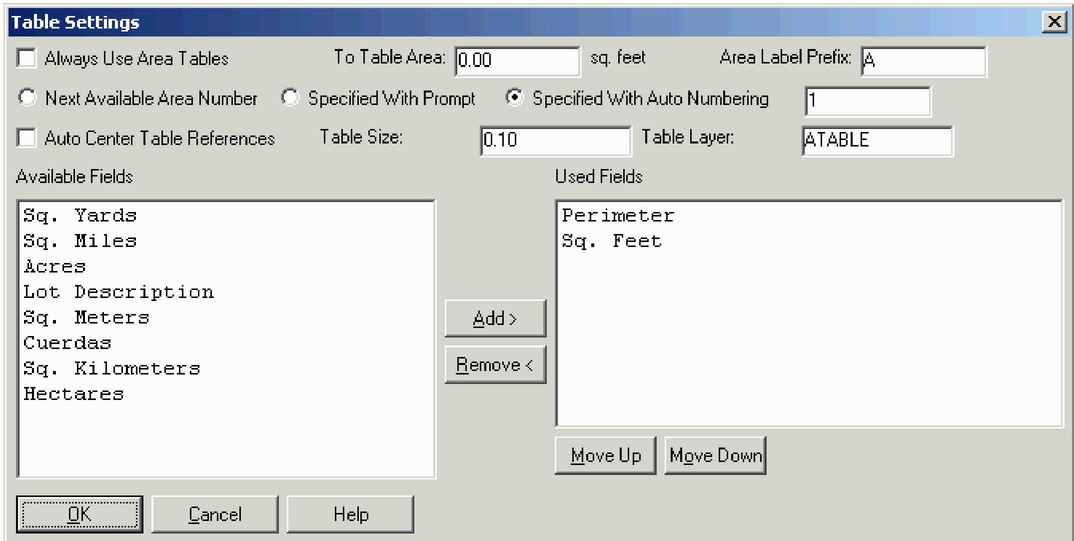


Area Defaults

This command allows you to specify default settings for area labeling. The top portion of the Area Defaults dialog lists ten area values. For each value, you can specify a value under the Order# column, a Prefix and a Suffix. The area label will include the values in the order specified. If an Order# is left blank, that value is not labeled. Both prefix and suffix controls are included here, although most area labeling uses the suffix, as in 1.25 Acres or 3.515 Hectares. But for those who prefer a prefix, as in Ac: 1.25, this routine can create that area labeling style automatically. Keep in mind that changes in Area Defaults, if changed in the Area pulldown menu, only apply to that work session. If changed within the Configure command, the changes apply to all new work sessions as well.



Sequence: The top portion of the dialog lists ten area values. For each value, you can specify a value under the Order# column, a Prefix and a Suffix. The area label will include the values in the order specified. If an Order# is left blank, that value is not labeled.
Precision for Square Units Labels: Choose precision level for labeling Square Feet, Square Yards, Square Miles, Square Meters, and Square Kilometers.
Precision for Other Area Labels: Choose precision level for labeling Acres, Cuerdas, and Hectares.
Precision for Inverse with Area: Choose precision level when using Inverse with Area.
Label Area with +/-: This allows you to display + or - in the Prefix or Suffix of the area labels, or choose None.
Use Commas in Labels: This allows you to use commas in the area labels.
Label Both Feet & Meters for Inverse with Area: When this option is turned ON, both feet and meters will be shown in the *Inverse with Area* report.
Layer for area text: This allows you to assign a layer for the area text.
Style for area text: This allows you to set a text style for the area labels.
Area text size scaler: This value is multiplied by the horizontal scale to obtain the actual text size.
Max gap to join (Area by Lines and Arcs): You use this option during *Area by Lines & Arcs* command. When connecting lines and arcs that define the perimeter, the program will join endpoints if the distance between the two points is less than the specified gap. Otherwise the program will report an error and will not report an area.
Different Radius Tolerance: You use this option to check the difference between the PC-Radius and PT-Radius on curves. If the difference between these distances is greater than this tolerance, an accurate area calculation cannot occur and the command displays a warning. This setting is used in Inverse, Inverse With Area, Hinged Area and Sliding Side Area.
Prefix/Suffix: Although most area labeling uses the suffix as in 1.25 Acres or 3.515 Hectares, those who prefer a prefix as in Ac: 1.25 can create that area labeling style automatically. This control is shown in the dialog below. Keep in mind that changes in Area Defaults, if changed in the Area pulldown menu, only apply to that work session. If changed within the Configure command, the changes apply to all new work sessions as well.
Load/Save: These buttons save and recall all the Area Default settings to a .ARS settings file.



The **Area Table Settings** button from the main dialog brings up the Table Settings shown above. The area table option puts the area data in a table that is typically drawn outside the area and contains area data for multiple areas. Each row in the table has the data for one area and includes a reference number. The reference number is also labeled inside area.

Always Use Area Tables will label all the areas with the table regardless of the area size.

The **To Table Area** will only use the area table for areas less than the specified amount. If you don't want to use area tables, set this amount to zero.

The **Area Label Prefix** is used for the reference number.

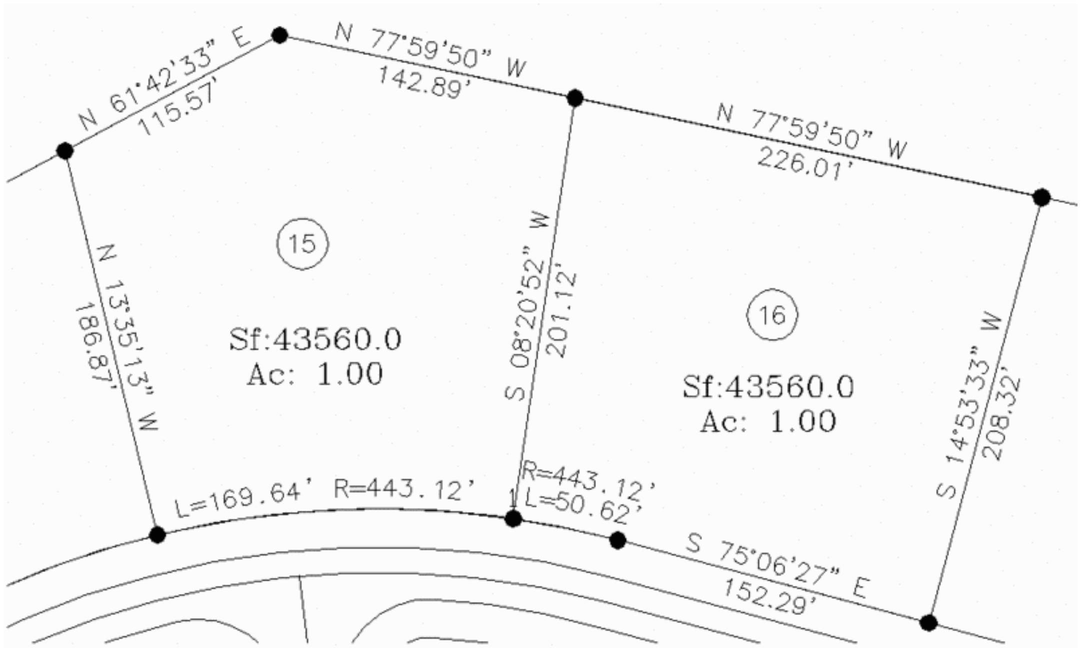
There are three different methods for setting the reference number: **Next Available** will automatically use the lowest available number. **Specified With Prompt** will prompt you for a number for each area. **Specified with Auto Numbering** will automatically use the lowest available number starting with the specified number.

Auto Center Table References will automatically place the reference number in the center of the area. Otherwise you will be prompted to pick each label location.

Table Size is a scaler that is multiplied by the current horizontal scale to size the table text size.

Table Layer is the layer name for the area table entities.

The **Available/Used Fields** allow you to specified which fields to include in the table and their order.



The results of the using prefix with square feet and acres

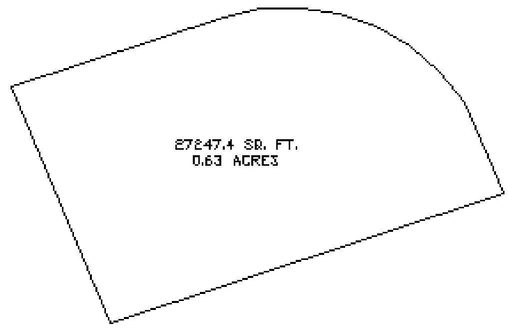
Pulldown Menu Location: Area/Layout
Keyboard Command: defarea
Prerequisite: None
File Name: \lsp\defarea.lsp, \lsp\scadarea.dcl

Area By Inverse

This command generates a report of the bearing and horizontal distance between a series of points, and calculates the area of the closed figure defined by the points. Curve data can also be entered and reported. The points can be either picked on the screen, or entered by point number. You can also enter a range of point numbers (i.e. 1-9). The closure is reported using the total distance inversed, and the difference between the starting and ending points, as the closure error. There is an option in Area Label Defaults to report the distances in both feet and meters. The area can be labeled in the drawing using the settings from the *Area Label Defaults* command. If you don't want to label the area, press Enter at the pick label point prompt. This command creates a polyline of the figure which can be erased or kept in the drawing.

Prompts

Station/<Pick Starting point or point number>: *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *R* for radius
Radius point number or pick point: *pick a point*
Curve direction [Left/<Right>]? *press Enter*
Pick End of Arc or point number (U-Undo,Enter to end): *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *pick a point*




```
 Inverse with Area 05/19/2005 13:07
CRD File> c:\scad2006\data\newplat.crd
PNTNO  BEARING       DISTANCE  NORTHING  EASTING  STATION  DESC
903                            4940.73   2490.40  0.00     StartPt
       S 48&#176;43'58'' W 136.21
904                            4850.89   2388.01  136.21
       S 13&#176;07'04'' E 155.56
905                            4699.39   2423.32  291.77
       S 73&#176;55'04'' E 165.34
906                            4653.59   2582.19  457.11
   RADIUS: 111.45   CHORD: 204.07     DEGREE: 05&#176;08'28''   DIR: LEFT
   LENGTH: 257.86   DELTA: 132&#176;33'53  TANGENT: 253.67
   CHORD BRG: N 39&#176;47'59'' E   RAD-IN: N 16&#176;04'56'' E   RAD-OUT: S 63&#176;31'03'' W
   RADIUS PNTNO: 907 N: 4760.67  E: 2613.06
908                            4810.37   2712.82  714.97   IP
       N 26&#176;28'57'' W 125.87
909                            4923.03   2656.69  840.84
       N 83&#176;55'30'' W 167.23
903                            4940.73   2490.40  1008.07  StartPt
Closure Error Distance> 0.0000
Total Distance Inversed> 1008.07
AREA: 74664.6 SQ METERS
```

Point number (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *press Enter*
SQ. FEET: 27247.4 SQ. YARDS: 3027.5 SQ. MILES: 0.0
ACRES: 0.63 PERIMETER: 668.35
Pick area label centering point: *pick a point*
Erase Polyline Yes/No <Yes>: *press Enter* The command plots a polyline that represents the figure you defined if you want to keep the polyline respond with No.

Pulldown Menu Location: Area/Layout
Keyboard Command: ia
Prerequisite: None
File Name: \lsp\ia.lsp

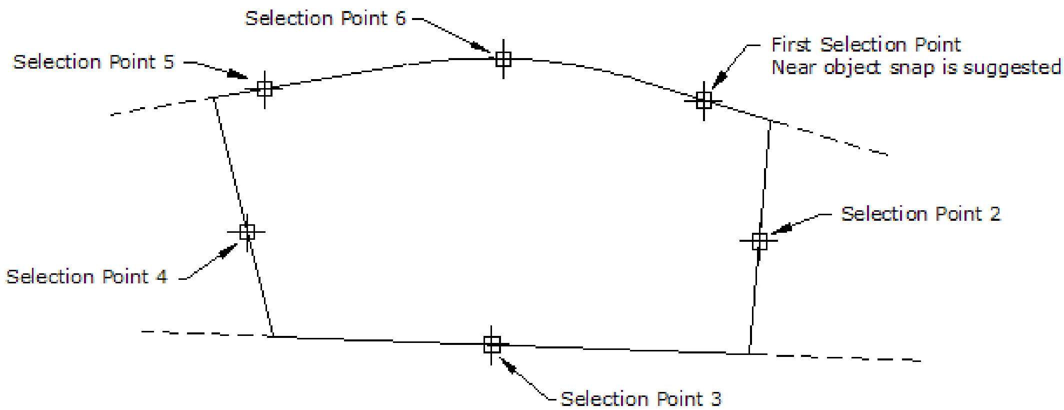
Area by Lines & Arcs

This command allows you to calculate the area of a perimeter or lot defined by lines, arcs, or polylines. Default settings for this command are set in *Area Defaults*. One of these settings is Max gap to join. If there is a gap greater than this value, the area is not reported, and the program will show where the gap is with a temporary X symbol. The area data shows up on the text screen. You can then choose to plot the area information to the drawing, or, by

hitting *Enter*, just read it from the text screen.

Prompts

Select lines & arcs or polylines of perimeter for area calculation.
Select Objects: *select lines and arcs or polylines*
Lines and arcs are then joined together and the area calculated.
Pick area label centering point (Enter for none): *pick a point*
The area is then plotted at the point selected.



Pulldown Menu Location: Area/Layout
Keyboard Command: joinarea
Prerequisite: Lines, arcs, or polylines to define the area
File Name: \lsp\joinarea.lsp

Area by Interior Point

This command calculates and labels the area of the perimeter surrounding a picked interior point. The *Boundary Polyline* command is used to find the perimeter. Generally, this command will only work on closed or overlapping objects. Use *Area by Lines & Arcs* for other applications.

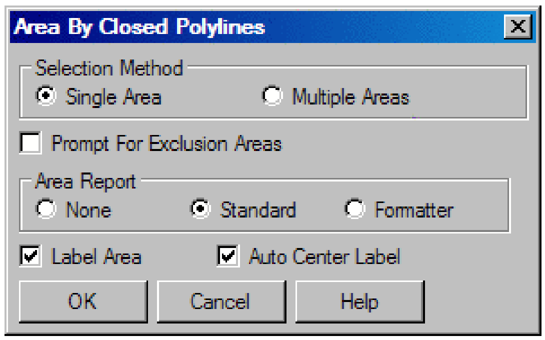
Prompts

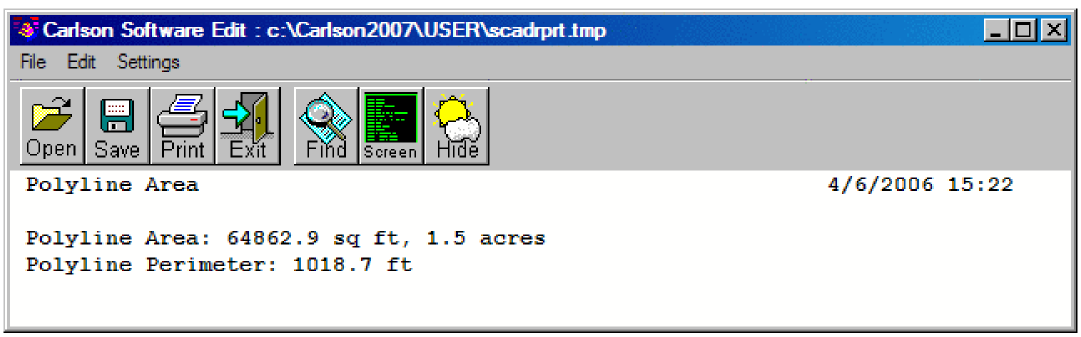
Pick point inside area perimeter: *pick a point*
Pick area label centering point (Enter for none): *pick a point*
The area is then plotted at the point selected.

Pulldown Menu Location: Area/Layout
Keyboard Command: ptarea
Prerequisite: Set Area Label Defaults
File Name: \lsp\ptarea.lsp

Area by Closed Polylines

This command will calculate and report the area of single area and multiple area closed polylines. Area by Closed Polyline will also automatically find special Carlson attributes attached to the polyline, in addition to capturing the area itself. These attributes will appear in the report, which can be the standard report or which can be presented in the Report Formatter, which itself links to Excel and Access. For example, property names and owner names, as applied to a polyline using the Mine modules, will report out automatically using Area by Closed Polyline. The command "Draw Lots from File..." will apply "extended entity data" to the lot polylines, which includes the lot name, and this will also report out when using Area by Closed Polyline. In addition, lot names, or any interior text whatsoever, can be captured and included in the report. The plot of the area on-screen can be canceled if only the report is desired.





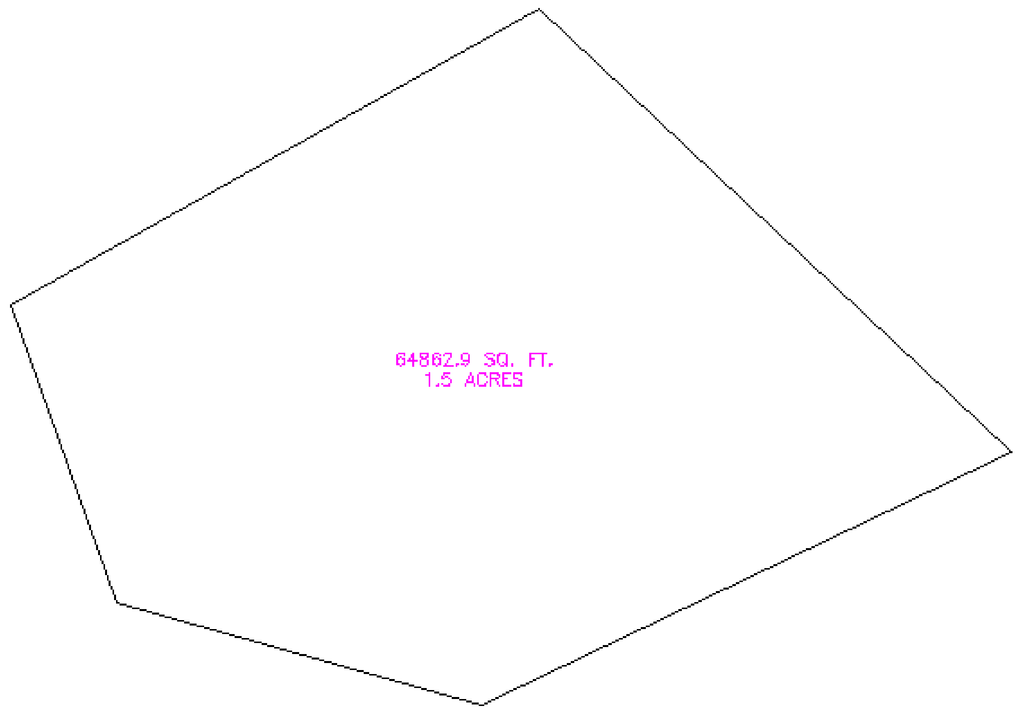
Prompts

Select Area Polyline: *select the area polyline*
SQ. FEET: 64862.9 SQ. YARDS: 7207.0 SQ. MILES: 0.0
ACRES: 1.5 PERIMETER: 1018.7
Pick area label centering point (Enter for none): *pick a location*

When additional interior text is selected, the standard report will include that text:

Polyline Area 11/17/2004 12:49
Polyline Area: 43560.0 sq ft, 1.00 acres
Polyline Perimeter: 838.35 ft
Text: 16 Sf: 43560.0; Ac: 1.00

In this case, the "16" refers to Lot 16, and appears in the report because the lot number and existing area labeling were selected along with the polyline for the lot.



Pulldown Menu Location: Area/Layout
Keyboard Command: plarea
Prerequisite: Set Area Label Defaults
File Name: \lsp\ptarea.lsp

Tag Area Descriptions

This command is one of three routines used to assign and report area descriptions. This command adds a tag to the closed area. The existing polyline must be closed.

Prompts

Select polyline for area description: *pick a polyline*
Area description <AREA1>:

Pulldown Menu Location: Area/Layout > Area Descriptions
Keyboard Command: tag_area_desc
Prerequisite: A closed polyline.
File Name: \lsp\areaac.lsp

Identify Area Descriptions

This command is one of three routines used to assign and report area descriptions. This command adds a tag to the closed area. Therefore, the existing polyline must be closed.

Prompts

Select polyline for area description: *pick a polyline*
Area description <AREA1>:

Pulldown Menu Location: Area/Layout > Area Descriptions
Keyboard Command: tag_area_desc
Prerequisite: A closed polyline.
File Name: \lsp\areaac.lsp

Untag Area Descriptions

This command is one of three routines used to assign and report area descriptions. This command adds a tag to the closed area. The existing polyline must be closed.

Prompts

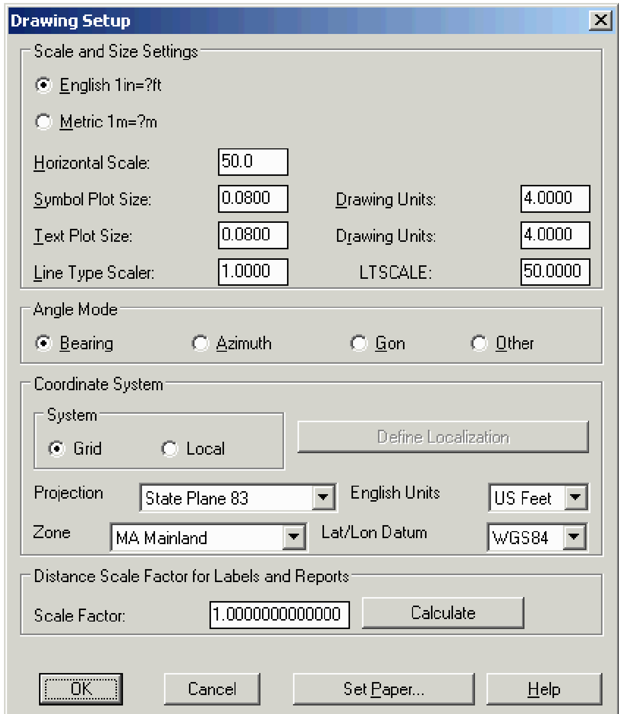
Select polyline for area description: *pick a polyline*
Area description <AREA1>:

Pulldown Menu Location: Area/Layout > Area Descriptions
Keyboard Command: tag_area_desc
Prerequisite: A closed polyline.
File Name: \lsp\areaac.lsp

Settings Menu 15

Drawing Setup

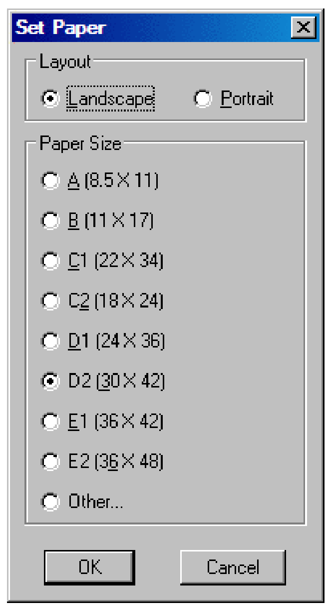
This command allows you to specify drawing parameters, including the plotting scale, size of symbols, label annotation size, and the angle mode.



- Specify **English 1in=?ft** or **Metric 1m=?m** as the unit mode to use. This affects the prompting and reports. When you are working on a drawing in English units, one unit equals one foot. In metric, one unit equals one meter.
- Specify the **Horizontal Scale** of the drawing. For example, if the horizontal scale is set to 50, then 1" = 50' is your drawing scale.
- The **Symbol Plot Size** value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The **Drawing Units** field shows the result of the Symbol Plot Size value (the scaler) multiplied by the horizontal scale.
- The **Text Plot Size** value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Text Plot Size is not entered in Drawing Units. The **Drawing Units** field shows the result of the Text Plot Size value (the scaler) multiplied by the horizontal scale.
- The **Line Type Scaler** option sets the linetype scale by multiplying this scaler by the horizontal scale.
- **Angle Mode-Bearing** sets reporting to bearing mode for any of the inquiry commands. (Modifies the settings in the AutoCAD *UNITS* command.)
- **Angle Mode-Azimuth** sets reporting to north based azimuth mode for any of the inquiry commands. (Modifies the settings in the AutoCAD *UNITS* command.)
- **Angle Mode-Gon** sets reporting to gon mode for any of the inquiry commands. (Modifies the settings in the AutoCAD *UNITS* command.)
- **Angle Mode-Other** lets the user determine angle mode by using the AutoCAD *UNITS* command.

- **Coordinate System** is an optional setting to define the drawing coordinate system. The coordinate system settings are used in commands like List Points and Label Lat/Lon to report geodetic coordinates from the drawing coordinates. The Grid System setting applies to drawing coordinates that are in a grid projection system such as state plane coordinates. The Projection list selects the grid projection from the list of supported projections. Along with the Projection, there are selections for the zone and datum to use with the projection. When the drawing setup is in English mode, there is a projection setting for whether the feet are in US Feet or International Feet units. The Local System setting applies to all other coordinate system beside grid projections. The Define Localization button has settings to define the transformation from local coordinates to grid coordinates. With a localization defined, you can work in a drawing in local coordinates and still report lat/lon. The localization definition contains pairs of local and grid coordinates that define the transformation. See the section on Localization under the Coordinate File Utilities command for more information.
- **Distance Scale Factor for Labels and Reports** is used to show distances in a second system besides the drawing units. For example, this factor can be used to report distances in meters when the drawing is in feet, or it can be used to report grid distances when the drawings is in a ground coordinate system. This factor is applied in commands that have an option to label/report a second scaled distance such as the Inverse command and Annotate Defaults that applies to the angle/distance label routines. The scale factor can be entered directly into the edit box or calculated using the Calculate button which has feet-meters conversions as well as combined scale factor calculations for grid-ground factors. See the Scale Points command for more information on calculating the combined scale factor.

- The **Set Paper** button allows you to draw a rectangle on the screen that represents the edge of your paper. After you have set the horizontal scale, press the Set Paper button and the Set Paper dialog appears.



  - The **Layout** option lets you specify landscape or portrait paper orientation. Landscape layout is where the width of the page is greater than the height of the page. Portrait layout is the opposite.
  - The **Paper Size** option allows you to specify the paper size. The numbers in parenthesis represent drawing units and will be multiplied by the horizontal scale to determine the rectangle to be drawn. If you select the Other option, you will be prompted on the command line for the horizontal and vertical sizes of the paper.

Prompts (for Set Paper)

Pick or Type lower left corner point for border <(5000.00 5000.00 0.0)>: *pick a point*
Erase existing Set Paper boundary [<Yes>/No]? *Y* This prompt only appears if there is an existing paper boundary in this drawing.
Set Limits [Yes/<No>]? *Y* If you answer Yes to Set Limits, drawing limits are enabled, and AutoCAD restricts the coordinates you can enter to within the paper boundary. Drawing limits also determines the area of the drawing that can display grid dots, and the minimum area displayed by the Zoom All command on the View menu. To turn drawing limits off, type in LIMITS on the command line and set to Off.

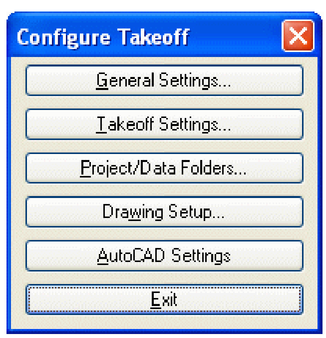
Drawing Setup also sets the AutoCAD dimension scale (DIMSCALE) and linetype scale (LTSCALE) to the Horizontal Scale.

Pulldown Menu Location: Settings
Keyboard Command: setup
Prerequisite: None
File Names: \lsp\survset.lsp, \lsp\scadenvr.dcl

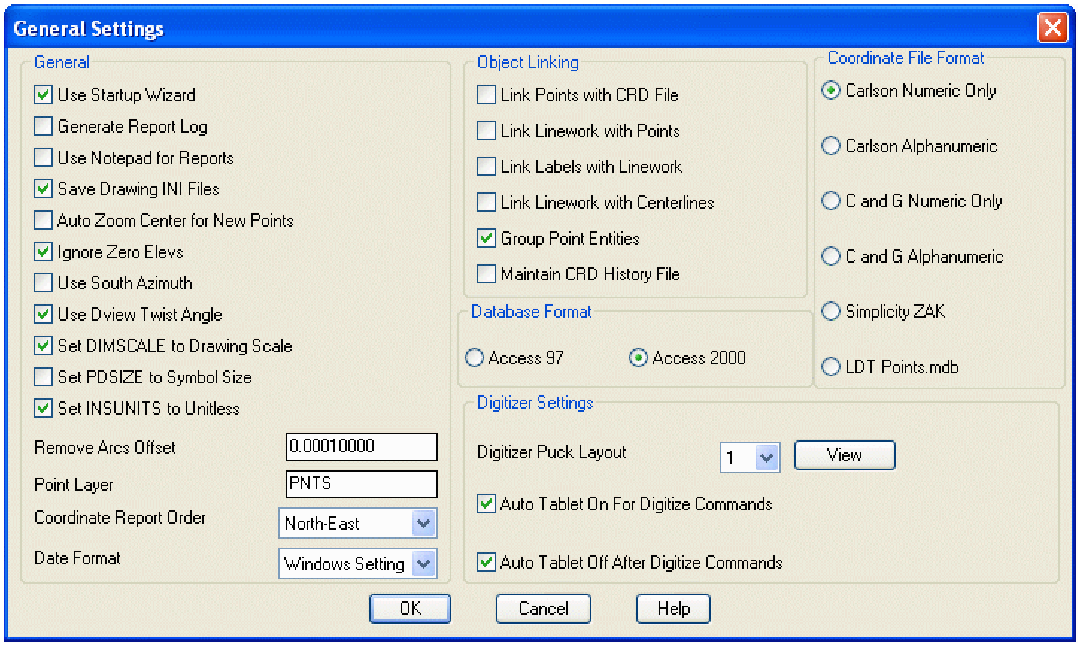
Configure

This command allows you to set the default settings that are used each time you start a new drawing or load an existing drawing. These settings are stored in *.ini files in the Carlson TakeOff directory. Configure restores the current drawing settings to these default settings.

1 In the Configure dialog box you choose between General Settings, Takeoff Module, Drawing Setup, and AutoCAD Settings.



2 In the General Settings dialog box you can set options for Carlson Takeoff.



A Under General, you can choose options relevant to points, angles, and start up.

- **Use Startup Wizard**: This option controls whether the wizard appears when you create a new drawing.

- **Generate Report Log**: This option allows output from several commands to be accumulated in a report buffer. Any report that is displayed in the standard report viewer is also added to the report log. While activated, the report log resides in the lower left corner of the desktop as a minimized title bar displaying the number of lines in the report buffer. To view the report log, pick the maximize icon on this title bar. You can edit the report log, save it to a file, or print it. To quickly turn the report log on and off, you can type REPORT at the command prompt, which toggles the report log on/off.

- **Use Notepad for Reports**: When this toggle is turned on, whenever a report is generated, it will appear in a Windows Notepad instead of the TakeOff Report Viewer.

- **Save Drawing INI Files**: This option creates an .ini file with the same name as the .dwg file to store the project data files for the drawing.

- **Put Data Files in DWG Directory**: This option sets the Data Path to the directory of the drawing. The Data Path is the default directory for data files such as the coordinate file (.crd).

- **Auto Zoom Center for New Points**: This option will zoom center on new points.

- **Ignore Zero Elevs**: This option causes entities with zero elevations to be excluded from calculations, etc.

- **Use South Azimuth**: This option allows you to use a south azimuth for calculations.

- **Use Dview Twist Angle**: This option keeps text horizontal to a twist screen view.

- **Set Dimscale to Drawing Scale**: This option sets the AutoCAD Dimscale setting to equal the current drawing scale.

- **Set PDSIZE to Symbol Size**: When checked, the system variable PDSIZE will be set to the same size as the symbol size that you set in Drawing Setup. PDSIZE controls the display size of AutoCAD point entities. Normally AutoCAD point entities are displayed as a dot, and the size does not apply. You may modify the point display type

by changing the system variable PDMODE. For example, if you set PDMODE to 64, point entities are displayed as a square regardless of the TakeOff symbol type used.

- **Remove Arcs Offset:** This makes arcs into straight lines if the arcs are smaller than the offset.

- **Point Layer**: You can assign a default layer name for points.

- **Date Format**: You can control the display of dates in Carlson TakeOff reports with this popdown menu. The default is 'Windows Setting' which allows you to control it with Windows Control Panel. Several other common formats are available.

- **Digitizer Puck Layout**: This option sets the layout for the digitizer puck keys. Pick the View button to see a graphic of the different layouts.

B Under Support Paths, you must determine paths for file allocation and retrieval.

- **MS Excel Path**: You determine the path for the *.exe file for MS Excel is located.
- **Coordinate Report Order**: You can choose whether coordinates are reported in northing-easting or easting-northing order.

C Under Object Linking, you can set reactors to the drawing entities.

- **Link Points with CRD File**: This option attaches a reactor to the point entities so that any change to the entities such as MOVE or ROTATE will update the coordinates in the coordinate file.
- **Link Linework with Points**: This option attaches reactors to line and polyline entities that are drawn by point number so that moving the points automatically moves the linework.
- **Link Labels with Linework**: This option applies to bearing/distance annotation. This link updates the annotation when the linework is modified.
- **Group Point Entities**: This option joins the three entities of a point (attribute block, symbol, node). For each point, selecting any one of these entities selects all three entities for the point.

D Under CRD File Pt# Format, you can set the point number format for coordinate files.

- **Carlson Numeric Only**: This is the default format upon installation. Point numbers cannot contain letters and must be in the range from 1 to 32767.
- **Carlson Alphanumeric**: This native Carlson format allows letters in the point numbers, and the point name can be up to 10 characters. Any combination of letters and numbers is acceptable.

• **C&G Numeric**: This format of the C&G division supports up to 5 digits, with a 65000 point limit.

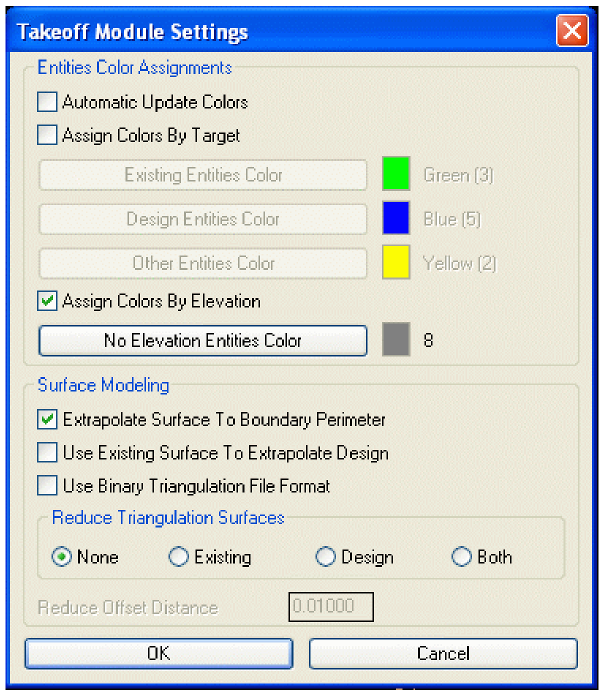
• **C&G Alphanumeric**: This format of the C&G division supports up to 10 characters, with no limit to the number of points.

• **Simplicity ZAK**: This format is the one used by Simplicity Systems programs. The file uses a Microsoft Access database format.

• **LDD Points.mdb**: This is a Microsoft Access database used by Autodesk Land Desktop. The file is typically named "points.mdb" and is found in a projects \Cogo directory. The number limitation is established by the database structure, but is frequently numeric and allows unlimited point numbers.

E Under Database Format you choose between Microsoft Access 97 or 2000 format. This only applies to new .MDB files created by Takeoff.

3 Takeoff Settings



• **Automatic Update Colors:** This refreshes colors in your drawing as they change: i.e. elevating entities, setting layers for different Targets, etc. If your drawing is very large and is slow to automatically refresh you may want to toggle this off and use the **Update Colors For Set Elevations** command under View when you want/need to see the color changes.

• **Assign Colors By Target**: This option allows you to set the Existing, Design, and Other layers to any color you define.

• **Assign Colors By Elevation**: This option allows you to set the color for entities needing elevations.

• **Extrapolate Surface To Boundary Perimeter**: When this is check ON surfaces are extended and volumes are calculated out to your boundary perimeter. When this is checked OFF surfaces and calculations end at the extents

of your design data.

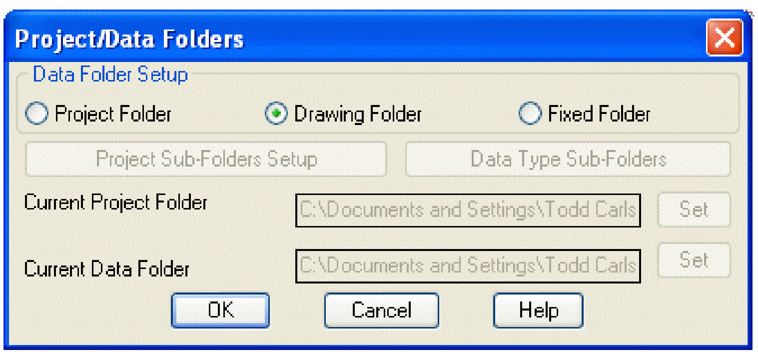
• **Use Existing Surface To Extrapolate Design**: When this is checked ON surfaces and volumes are calculated to the extents of your existing data.

• **Use Binary Triangulation File Format**: This option sets the format for the surface model files as either binary or ASCII. The binary format has a .tin file name extension and loads about twice as fast and has about 50% less file size than ASCII. The ASCII format has a .flt extension and is the legacy format used by other Carlson products and Softdesk.

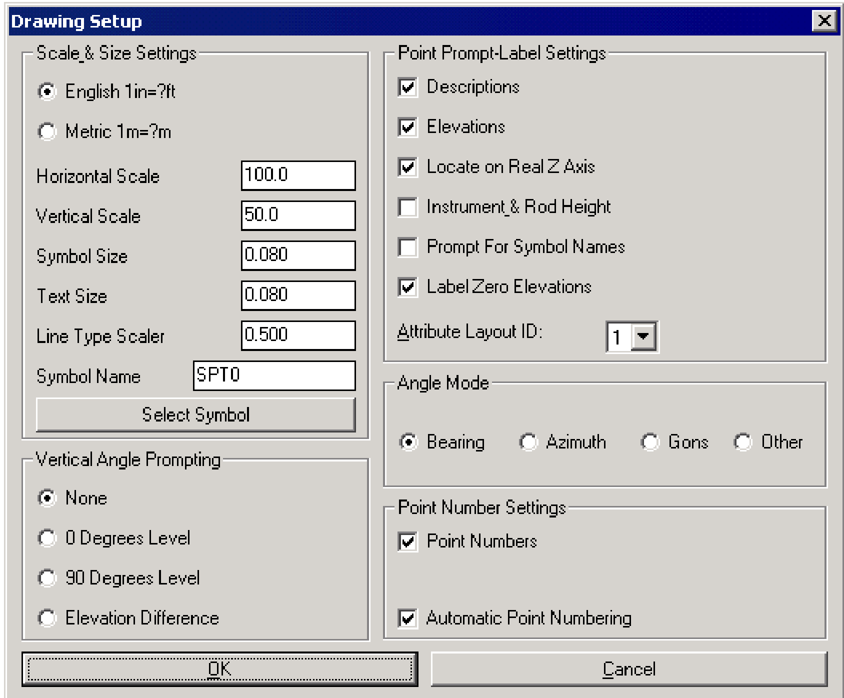
• **Reduce Triangulation Surfaces**: This causes edges within the selected surface Tin mesh to be collapsed to reduce the number of triangles, edges, and points within the mesh while having a minimal impact on the overall shape of the mesh.

Reduction Offset Distance: This setting is used by the Reduce Triangulation Surfaces command to set the reduction tolerance. Specify the maximum average distance that any point can be moved outside of the plane of any triangle that connects to that point. Values might range from .01 to .1 for most purposes.

4 This dialog box allows you to set up your. **Data Path**. You determine can where data files (∗.crd, ∗.tin, etc.) are stored. Either in defined folders (Project Folder), the same place where your drawing (.dwg) is stored, which is the default and recommended (Drawing Folder), or in the same folder no matter which drawing you are working on (Fixed Folder).



5 In the Drawing Setup dialog box you have options for setting drawing parameters, including the plotting scale, size of symbols, label annotation size, and the drawing mode.



A Under Scale and Size Settings, you can determine scale and size of drawing entities.

• **English 1in=?ft**: This option tells the program which unit mode to use. This affects the prompting and reports. When you are working on a drawing in English units, one unit equals one foot.

• **Metric 1m=?m**: This option sets the metric scale to meters only.

• **Horizontal Scale**: This option allows you to set the horizontal scale of the drawing. For example, if the horizontal scale is set to 50, then 1" = 50' in your drawing.

• **Vertical Scale**: This option allows you to set the vertical scale of the drawing.

• **Symbol Size**: . This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Symbol Plot Size is not entered in Drawing Units

• **Text Size**: This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Text Plot Size is not entered in Drawing Units.

• **Line Type Scaler**: This option sets the linetype scale by multiplying this scaler by the horizontal scale.

• **Symbol Name**: This option allows you to set the default symbol name for points.

• **Select Symbol**: Click this button to graphically select the default symbol.

B Under Point Prompt-Label Settings, select the options that determine how the points are to be labeled and how you will be prompted for point entry.

• **Descriptions**: Determines whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.

• **Elevations**: Sets prompting and labeling for point elevations.

• **Locate on Real Z Axis**: Switches between locating points at zero elevation and at the actual stored elevations.

• **Instrument & Rod Height**: Turns on prompting for instrument and rod heights when creating points.

• **Prompt for Symbol Name**: When checked, the program will prompt for a symbol name as each point is drawn. Otherwise, the default symbol name set in this dialog box will be used.

• **Attribute Layout ID**: Controls the location of the point number, elevation, and description. These attribute layouts are defined in drawings that are stored in the TakeOff Support directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then edit the associated SRVPNO drawing.

C Under Angle Mode, you determine how angles are entered and displayed.

• **Bearing**: This option sets reporting to bearing mode for any of the Inquiry commands.

• **Azimuth**: This option sets reporting to north based azimuth mode for any of the Inquiry commands.

• **Gon**: This option sets reporting to gon mode for any of the Inquiry commands.

• **Other**: Allows you to set a custom angle mode by using the Units Control command (described later in this chapter).

D Under Vertical Angle Mode select an option to determine how the vertical angle is calculated. Vertical Angle Prompt applies to creating points with commands such as Traverse.

• **None**: The vertical angle will not be used to calculate point elevations.

• **0 Degrees Level**: The vertical angle is used to calculate elevation and horizontal distance.

• **90 Degrees Level**: The zenith angle is used to calculate elevation and horizontal distance.

• **Elevation Difference**: Use the elevation difference to calculate the elevation.

E Under Point Number Settings, select the options that determine whether you will be prompted for point numbers by the commands that locate points.

• **Point Numbers**: When this toggle is checked on, points that are inserted by TakeOff commands are shown with a point number, and a coordinate is stored in the current coordinate (.crd) file. When this toggle is off, points are shown with no point number plotted and no coordinate is stored in the current coordinate (.CRD) file.

• **Automatic Point Numbering**: If this toggle is checked on, commands that locate a point will automatically insert a point number for each point drawn on the screen. If Automatic Point Numbering toggle is off, commands that locate a point will prompt for a point number.

5 AutoCAD Settings: This command allows the user to set up a number of options, pertaining to the directory file system, color combinations for the display screen, text fonts, workspace utitlities like shortcut keys etc. **Keyboard Command**: _preferences

Prerequisites: None

Keyboard Command: CONFIG_SCAD

Edit Symbol Library

Function

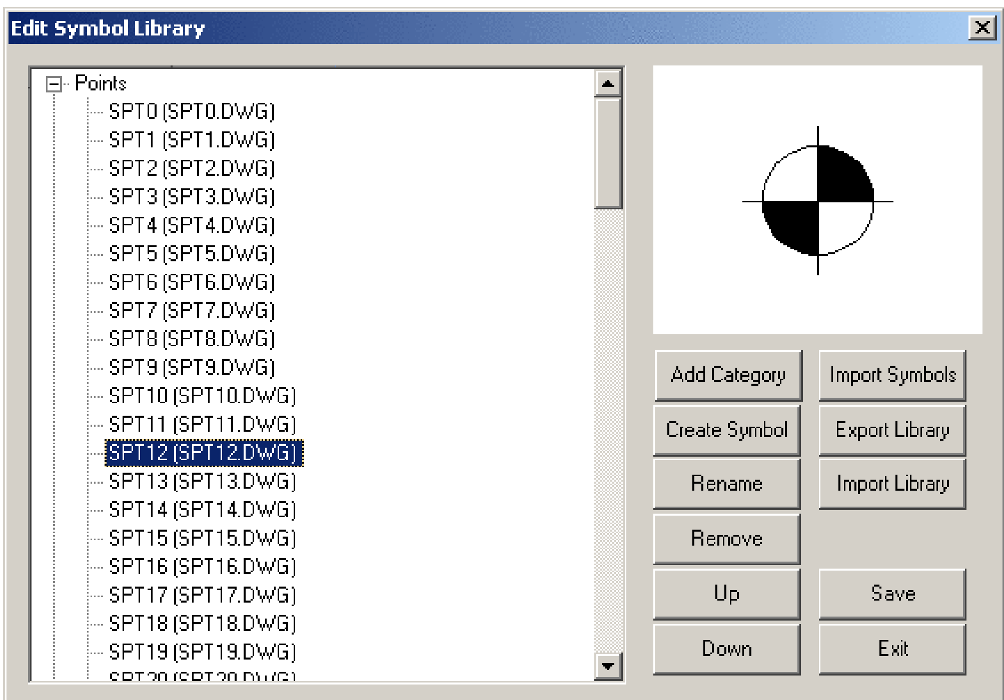
This command allows you to customize the symbol library.

Categories are a way for grouping symbols by type for your own convenience in symbol selection. A new category is added by clicking on the "Add Category" button. An edit field then appears in the tree view on the left and waits for you to enter the category name. The input is finished by pressing the Enter key.

The category may be populated by creating a new symbol from selected entities in the drawing, by specifying drawing (.DWG) files, or by moving existing symbols from one category to another.

To create a new symbol, open a drawing which has the entities to be used in the symbol. The symbol should be drawn at unit size (scale 1:1) because Carlson will scale the symbol by the current drawing scale when the symbol is used. Highlight the category for the symbol and click on the "Create Symbol" button. A dialog appears for entering the new symbol name. Next, specify the file name for the symbol. The file name has a .DWG extension and would usually reside in the Carlson SUP directory, but you may use another path. Then the program will prompt you to select the entities from the drawing for the symbol. An insertion point for the symbol must also be picked.

The "Import Symbols" button brings up a file selection dialog which allows you to select multiple files to be added to the current category (to select multiple files use Shift or Control keys along with the mouse). If the files you select are not in the Carlson SUP directory, the program will offer an option of copying them there. There are also Import Library and Export Library buttons.



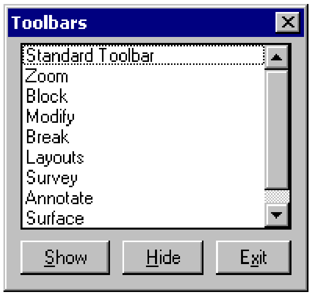
By default, the symbol description is the same as file name. The description for the symbol or category name may be changed by highlighting that name and clicking on "Rename" button, the name being edited is then placed into edit mode. To move a symbol into a different category, select the symbol to be moved on the tree and click an "Up" or "Down" button as many times a needed to reach the desired category. The symbols are sorted alphabetically within each category, while categories are remaining in the order placed to allow the more frequently accessed categories be on top.

Note: The symbol library is stored in an ASCII file named symbols.dta in the Carlson \USER directory.

Pulldown Menu Location: Settings
Keyboard Command: editptsym
Prerequisite: None
File Names: \lsp\scadutil.arx, \user\symbols.dta

Toolbars

This command allows you to display and hide toolbars. Click on a toolbar name and press the Show or Hide button.



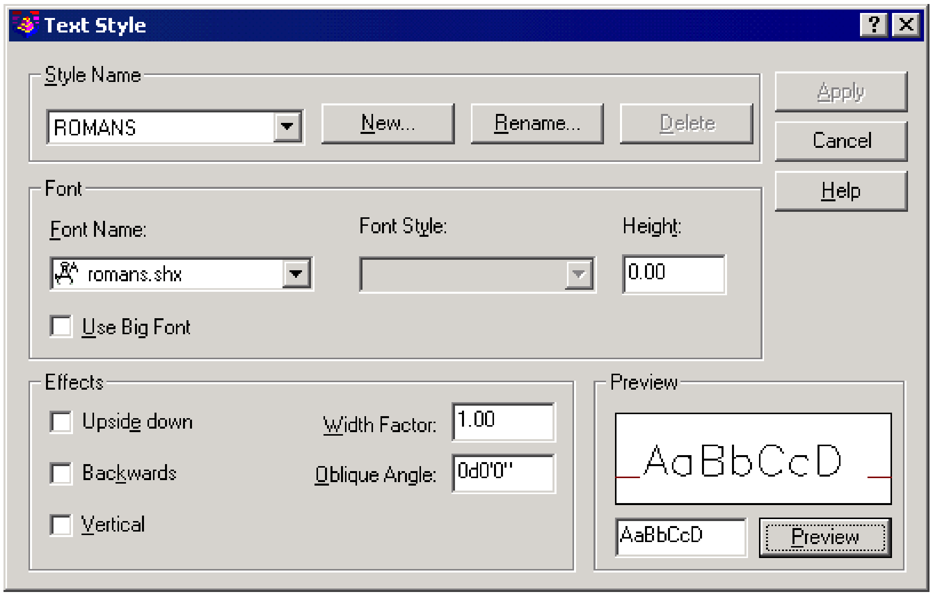
- **Show**: Turns on the selected toolbar. If the toolbar is already visible, then this does nothing.
- **Hide**: Turns off the selected toolbar. If the toolbars is already hidden, then this does nothing. If the toolbar is floating, you can also turn it off by clicking the x in the upper right corner.
- **Exit**: Exits this command

Prerequisite: None

Keyboard Command: TBARCFG

Text Style

This command creates or modifies named styles and sets the current style for text in your drawing



1 Under Style Name, you can display text style names, add new styles, and rename and delete existing styles. The list contains defined style names and displays the current style by default. To change the current style, select another style from the list, or choose New to create a new style.

- **New**: This option displays the New Text Style dialog box and automatically supplies the name "stylen" (where n is the number of the supplied style) for the current settings. You can accept the default or enter a name and choose OK to apply the current style settings to the new style name.

- **Rename**: This option displays the Rename Text Style dialog box. The text style listed is renamed when you enter a new name and choose OK.
- **Delete**: This option deletes a text style. Select a name from the list to make it current, and then choose Delete.

2 Under Font you can change the style's font.

- **Font Name**: This field lists the font family name for all registered TrueType fonts and all compiled shape (SHX) fonts in the TakeOff Fonts directory. When you select a name from the list, the program reads the file for the specified font. The file's character definitions are loaded automatically unless the file is already in use by another text style. You can define several styles that use the same font.
- **Font Style**: This field specifies font character formatting, such as italic, bold, or regular. When Use Big Font is selected, this option changes to Big Font Name and is used to select a Big Font file name.
- **Height**: This field sets the text height based on the value you enter. If you enter 0.0, the program prompts for the text height each time you enter text using this style. Entering a height greater than 0.0 sets the text height for this style. TrueType fonts can be displayed at a smaller height than SHX fonts with the same height setting. The text height you specify may not be accurately represented by uppercase letters in TrueType fonts supplied with TakeOff.
- **Use Big Font**: This option specifies an Asian-language Big Font file. Use Big Font is available only if you specify an SHX file under Font Name. Only SHX files are valid file types for creating Big Fonts.

3 Under Effects, you modify characteristics of the font, such as its height, width factor, and obliquing angle and whether it is displayed upside down, backwards, or vertically aligned. TrueType fonts using the effects described in this section might appear bold on the screen. Onscreen appearance has no effect on plotted output. Fonts are plotted as specified by applied character formatting.

- **Upside Down**: This option displays the characters upside down.
- **Backwards**: This option displays the characters backwards.
- **Vertical**: This option displays the characters aligned vertically. Vertical is available only if the selected font supports dual orientation. Vertical orientation is not available for TrueType fonts.
- **Width Factor**: This option sets the character spacing. Entering a value less than 1.0 condenses the text. Entering a value greater than 1.0 expands it.
- **Oblique Angle**: This option sets the obliquing angle of the text. Entering a value between -85 and 85 makes the text oblique.

4 Under Preview, you can display sample text that changes dynamically as you change fonts and modify the effects. To change the sample text, enter characters in the box below the character preview image.

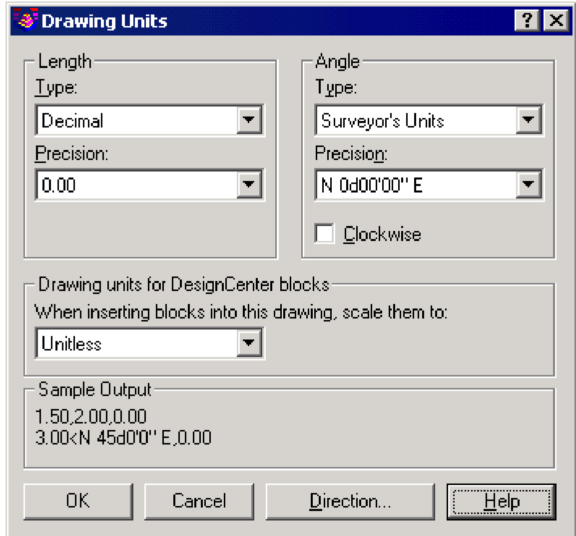
- **Preview**: This field updates the sample text in the character preview image according to any changes you've made in the dialog box. Height has no effect in the character preview image because a very large text height might show little or no text.

Prerequisite: None

Keyboard Command: STYLE

Units Control

The Drawing Units dialog box controls coordinate and angle display formats and determines precision.



1 Under Length, you specify the current unit of measurement and the precision for the current units.

- **Type**: This field sets the current format for units of measure. The values include Architectural, Decimal, Engineering, Fractional, and Scientific. The Engineering and Architectural formats produce feet-and-inches displays and assume that each drawing unit represents one inch. The other formats can represent any real-world unit.

- **Precision**: This field sets the number of decimal places for the current units display.

2 Under Angle you specify the current angle format and the precision for the current angle display.

- **Type**: This field sets the current angle format.

- **Precision**: This field sets the precision for the current angle display.

TakeOff uses the following conventions for the various angle measures: decimal degrees appear as decimal numbers, grads appear with a lowercase g suffix, and radians appear with a lowercase r suffix. The degrees/minutes/seconds format uses d for degrees, ' for minutes, and " for seconds, for example:

123d45'56.7"

Surveyor's units show angles as bearings, using N or S for north or south, degrees/minutes/seconds for how far east or west the angle is from direct north or south, and E or W for east or west, for example:

N 45d0'0" E

The angle is always less than 90 degrees and is displayed in the degrees/minutes/seconds format. If the angle is precisely north, south, east, or west, only the single letter representing the compass point is displayed.

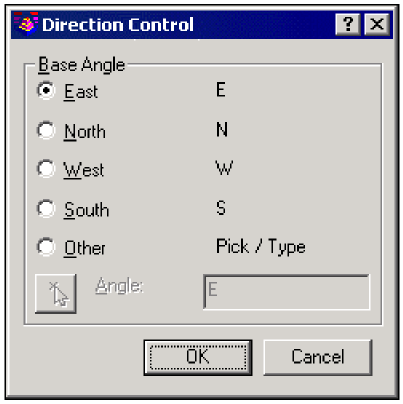
- **Clockwise**: This option calculates positive angles in the clockwise direction. The default direction for positive angles is counterclockwise.

When the program prompts for an angle, you can point in the desired direction or enter an angle regardless of the setting specified for Clockwise.

3 Under Drawing Units for TakeOff DesignCenter blocks, you can control the unit of measurement used for block insertions. A block created in units that differ from the units specified in this option is scaled and inserted in the specified units. Select Unitless to insert the block as is and not scale the block to match the specified units. Source content units and Target drawing units settings in the User Preferences tab of the Options dialog box under the Settings menu are used when Insert Units are not defined.

4 Sample Output displays an example of the current settings for units and angles.

Direction displays the Direction Control dialog box described below.



A The Base Angle determines where 0 degrees is located when the program calculates angles. The base angle sets the direction of the base angle. These options affect the entry of angles, object rotation angles, the display format, and the entry of polar, cylindrical, and spherical coordinates. Choose East, North, West, or South, or choose Other to indicate an alternative direction. The default direction for the zero angle is East. In TakeOff, the base angle is relative to the orientation of the user coordinate system.

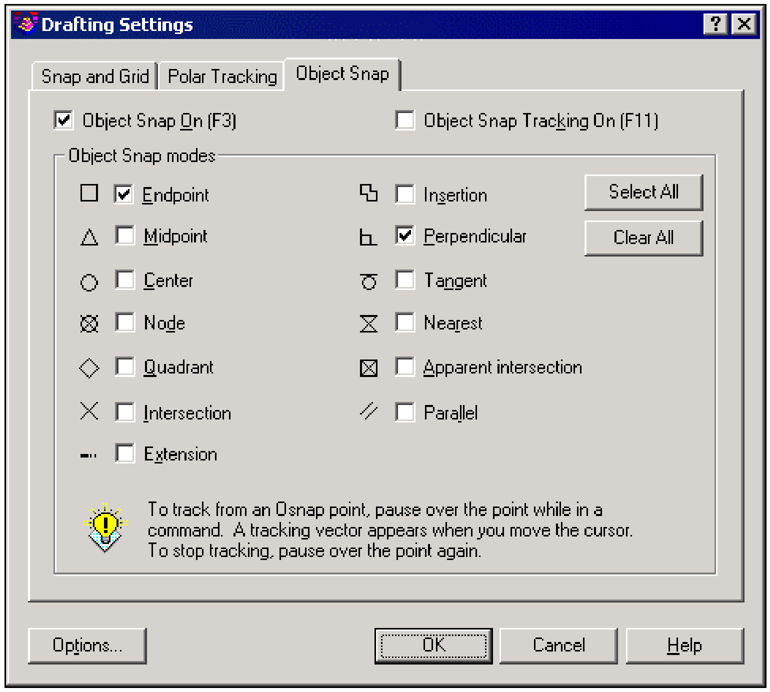
- **East**: Sets the base angle to east (default is zero degrees).
- **North**: Sets the base angle to 90 degrees north.
- **West**: Sets the base angle to 180 degrees west.
- **South**: Sets the base angle to 270 degrees south.
- **Other**: Sets a direction different from the points of the compass.
- **Angle**: Sets the angle. Available only when Other is selected.
- **Pick an Angle**: Uses the pointing device to define the angle based on the angle of an imaginary line connecting any two points you specify. Available only when Other is selected.

Prerequisite: None

Keyboard Command: UNITS

Object Snap

The Drafting Settings dialog box sets object snap modes.

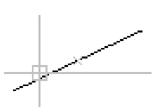


1 Under Object Snap, you set object snaps.

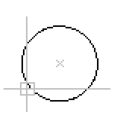
- **Object Snap On**: This option turns running object snaps on and off. The object snaps selected under Object Snap Modes are active while object snap is on. This setting is also controlled by the OSMODE system variable.
- **Object Snap Tracking On**: This option turns object snap tracking on and off. With object snap tracking the cursor can track along alignment paths based on other object snap points when specifying points in a command. To use object snap tracking, you must turn on one or more object snaps.

2 Under Object Snap Modes, you turn on running object snaps.

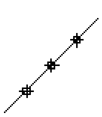
- **Endpoint**: Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray or to the closest corner of a trace, solid, or 3D face.
- **Midpoint**: Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, solid, spline, or xline.



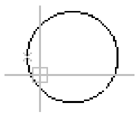
- **Center**: Snaps to the center of an arc, circle, ellipse, or elliptical arc.



- **Node**: Snaps to a point object.

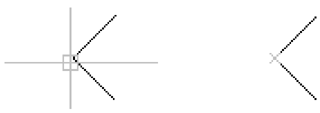


- **Quadrant**: Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.



- **Intersection**: Snaps to the intersection of an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline. Intersection snaps to the edges of regions and curves, but does not snap to the edges or corners of 3D solids.

Extended Intersection snaps to the imaginary intersection of two objects that would intersect if the objects were extended along their natural paths. Carlson TakeOff automatically turns on Extended Intersection when you select the Intersection object snap mode. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time. Intersection and Extended Intersection work with edges of regions and curves, but not with edges or corners of 3D solids.

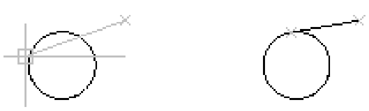


- **Extension**: Causes a temporary extension line to display when you pass the cursor over the endpoint of objects, so you can draw objects to and from points on the extension line.

- **Insertion**: Snaps to the insertion point of an attribute, a block, a shape, or text.

- **Perpendicular**: Snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, solid, spline, or xline. Carlson TakeOff automatically turns on Deferred Perpendicular snap mode when the object you are drawing requires you to complete more than one perpendicular snap. You can use a line, arc, circle, polyline, ray, xline, multiline, or 3D solid edge as an object from which to draw a perpendicular line. You can use Deferred Perpendicular to draw perpendicular lines between such objects. When the aperture box passes over a Deferred Perpendicular snap point, the program displays a Snaptip and marker.

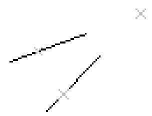
- **Tangent**: Snaps to the tangent of an arc, circle, ellipse, or elliptical arc. Carlson TakeOff automatically turns on Deferred Tangent snap mode when the object you are drawing requires you to complete more than one tangent snap. For example, you can use Deferred Tangent to draw a line that is tangent to two arcs, polyline arcs, or circles. When the aperture box passes over a Deferred Tangent snap point, the program displays a marker and Snaptip. If you use the From option in conjunction with the Tangent snap mode to draw objects other than lines from arcs or circles, the first point drawn is tangent to the arc or circle in relation to the last point selected in the drawing area.



- **Nearest**: Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, spline, or xline.

- **Apparent Intersection**: Apparent Intersection includes two separate snap modes: Apparent Intersection and Extended Apparent Intersection. You can also locate Intersection and Extended Intersection snap points while running Apparent Intersection object snap mode is on. Apparent Intersection snaps to the apparent intersection of two objects (arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline) that do not intersect in 3D space but may appear to intersect in the drawing display. Extended Apparent Intersection snaps to the imaginary intersection of two objects that would appear to intersect if the objects were extended along their natural paths. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time. Apparent and Extended Apparent Intersection work with edges of regions and curves but not with

edges or corners of 3D solids.



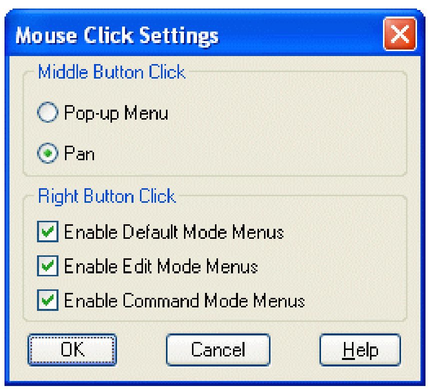
- **Parallel**: Draws a vector parallel to another object whenever Carlson TakeOff prompts you for the second point of a vector. After specifying the first point of a vector, if you move the cursor over a straight line segment of another object, the program acquires the point. When the path of the object you create is parallel to the line segment, the program displays an alignment path, which you can use to create the parallel object.
- **Clear All**: This option turns off all object snap modes.
- **Select All**: This option turns on all object snap modes.

Prerequisite: None

Keyboard Command: OSNAP

Mouse Click Settings

This command can be used to make custom mouse click preferences. The Middle Button Click applies to a 3-button mouse and chooses between using the middle button for real-time pan or to show an Object Snap pop-up menu. For the right mouse button, there are different levels of pop-up menus that can be activated. With all these menus off, the right button will be used like the Enter keyboard.



Keyboard Command: CLICKSET
Prerequisite: none

Set/Reset X-Hairs

Set X-Hairs sets the crosshairs either to align with the selected line or polyline or to a user-specified slope. *Reset X-Hairs* restores the crosshairs alignment to horizontal.

Pulldown Menu Location: Settings > Crosshairs
Keyboard Commands: setxhairs, resetxhairs
Prerequisite: Line entity
File Names: \lsp\xh.lsp, \lsp\rh.lsp

Set UCS to World

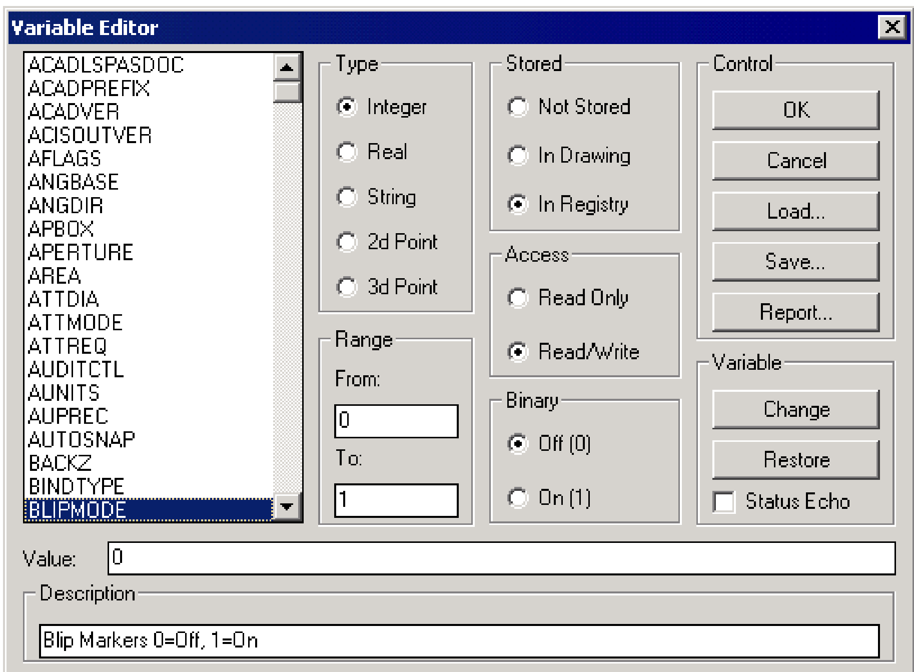
This command sets the UCS (user coordinate system) to the world coordinate system (WCS). Carlson TakeOff works exclusively in the world coordinate system and there is no way to change this setting. In AutoCAD, it is possible to change the coordinate system from WCS. If you receive a drawing in which the coordinate system is not set to world, use this command to restore the UCS.

Prerequisite: None

Keyboard Command: UCS_WORLD

Set Environment Variables

The AutoCAD engine stores the values for its operating environment and some of its commands in system variables. Each system variable has an associated type: integer, real, point, switch, or text string. This command allows you to list or change the values of system variables.



1 Dialog Fields

- **List Box**: Contains a list of the variables associated with the currently running version of AutoCAD. There are more items than will display on the list box, use the scroll bar to move up and down through the list. Picking on an item in the list box makes it the current item, causing the information about the item to be displayed, and can be affected by most of the edit commands explained below.
- **Edit Field**: When an item on the list box is picked, its current setting is displayed in the edit field. If you intend to make changes in an item, use standard editing procedures including the use of arrow keys and/or pointer movements to make changes. Once changes have been made, you must use the CHANGE options explained below to effect changes. Pressing enter at the edit field will have no effect on the item in the list. If the item selected is a read-only variable, the edit field will be grayed-out and will not allow input.
- **Description**: When an item on the list box is picked, its definition is referenced and displayed in this field. This can be a benefit in learning the uses of the assorted system variables. This is a display only field, so you can't change the description given.

2 Under Type Group, the type of variable will be displayed indicated by one of the radio buttons. Each of these types are explained below for your benefit. For additional information on variable types used by AutoCAD, obtain and consult a source of AutoCAD documentation.

- **Integer**: Defined as a whole number in the range from -32767 to +32768, no decimal value accepted.
- **Real**: Defined as a real number in the range from -1.797E+308 to +1.797E+308, with extreme decimal accuracy maintained.. Some real variables have a smaller range than previously stated.
- **String**: Defined as a sequential array of characters in the range from 0 to 65535 characters, with a range of ASCII (0-255). Numbers can be included in strings, even though they have no mathematical significance.
- **2D Point**: Defined as a list of two real numbers in the range from -1.797E+308 to +1.797E+308 separated by a comma, having extreme decimal accuracy maintained. Always maintain the X,Y format, one (and only one) comma must be used, separating the X and Y.
- **3D Point**: Defined as a list of three real numbers in the range from -1.797E+308 to +1.797E+308 separated by commas. While editing a 3D point, you must always maintain the X,Y,Z format, two (no less or no more), commas must used, separating the X and Y and Z values.

3 Under Range Group, the variable displayed will usually have a range displayed. The FROM value indicating the minimum, and the TO value being the maximum value accepted.

4 Under the Store Group, depending on the type of variable, AutoCAD may store the value in the drawing or the configuration file, or it may not be stored. Each of these types are explained below for your benefit.

- **Not Stored**: Some variables, such as PLATFORM and CDATE, are not stored because they are system interdependent.
- **In Drawing**: Most variables are stored in the drawing, making the drawing format more personal than just a database of objects. This allows you to open a drawing and have it behave just as though you had never left it.
- **In Config**: These are variables that remain the same regardless of the drawing opened. APERTURE and PICKBOX are just two examples of variables stored in the configuration file.

5 Under Access Group, depending on the type of variable, AutoCAD may not allow you to make changes to it. Each of these types are explained below.

- **Read Only**: Some variables, such as PLATFORM and CDATE, are read-only and therefore cannot be changed. Read-Only variables are marked and the edit field will be grayed indicating that you can't change the variable.
- **Read/Write**: Most variables are read/write and can be changed. These variables are marked and the edit field will be active so you can change the variable.

6 Under Binary Group, depending on the type of variable, the value may be off or on, yes or no. If the variable type is not binary, this group will be grayed out entirely.

- **Off (0)**: Indicate an off condition. Some variables, such as ATTREQ, are simply on or off toggles. You may change a binary item by clicking in this group to change the variable, or changing the value in the edit field.
- **On (1)**: Indicate an on condition. Binary variables are simply on or off toggles. Their range is from 0 to 1. You may change a binary item by clicking to change the variable, or changing the value in the edit field.

7 Control Buttons - These buttons are the main controls in the use of the Variable Editor. Each buttons purpose is explained below.

- **OK**: Used to accept the changes made during the variable editing process, returning you to the command prompt with changes in effect.
- **Cancel**: Used to cancel the changes made during the variable editing process, returning you to the command prompt without the changes in effect.

- **Load**: Used to load a saved set of system variables. This allows you to create a drawing, save the system variables, open a second drawing, and load those variables into that drawing. Read-only variables are skipped.
- **Save**: Used to save the current system variables to a disk file. All system variables are stored to the file, even those that are marked as read-only.
- **Print**: Used to print the current system variables. After choosing this option, you will prompted for an output filename, then the program will proceed to write the system variables to the file. This file can be loaded into any editor or word processor, edited and printed.

8 Variable Buttons - These buttons are used to control the changes in variables, while using the Variable Editor. Each buttons purpose is explained below.

- **Change**: Used to execute the changes typed into the edit field. You must use this button, simply pressing enter will not make the change.
- **Restore**: Used to cancel the changes typed into the edit field. If you make a mistake or change your mind while making changes in the edit field, press this button to restore the edit field to the value before editing.
- **Status**: Used to determine if the program will echo the status of changes being made to the command area. If this toggle is on, any changes made from the dialog will echo the change. Also if a stream of change commands is being read from a file, and the toggle is on, the changes taking place will be displayed.

Note: This command displays many more system variables than are found in Chapter 18, which contains a list of **supported** system variables. Modification of any system variable other than the supported ones found in Chapter 18 is done at your own risk, and may result in program errors requiring a re-installation of Carlson TakeOff.

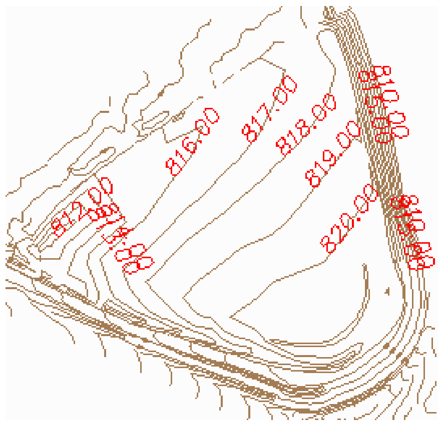
Prerequisite: None

Keyboard Command: VAREDIT

Display Menu 16

Existing Drawing

This command allows you to display all the entities on the layers that are grouped as part of the Existing Drawing.

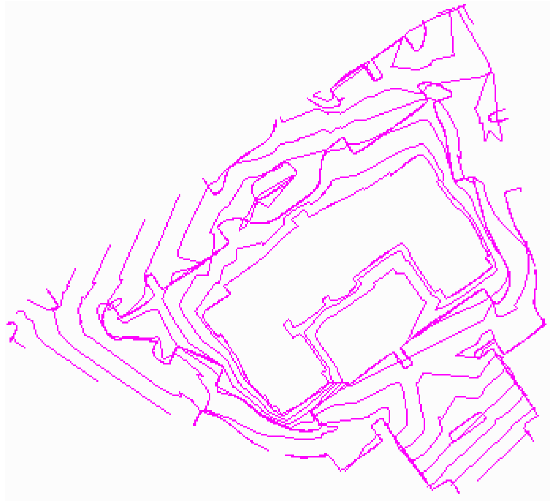


Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Existing Drawing is checked than the existing drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the existing drawing in isolation or in accordance to the other drawings.

Keyboard Command: set_display_exist_dwg
Prerequisite: Define Layer Target/Material/Subgrade

Existing Contours

This command displays all the contours that represent the existing surface (For contouring options see Display Options). Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

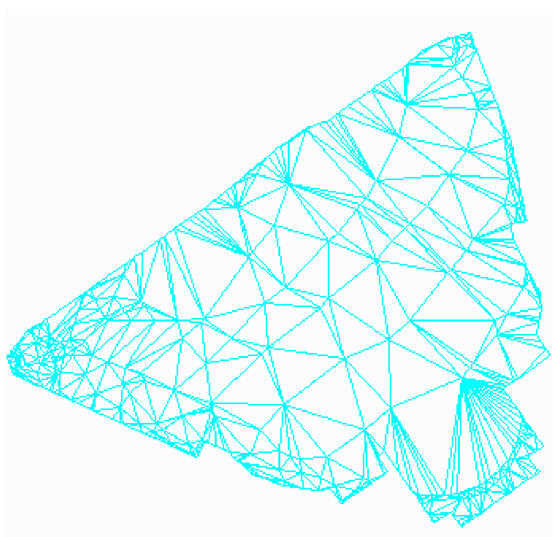


When Existing Contours is checked than all the contours for the existing surface will be displayed. If it is not checked they will not be displayed.

Prerequisite: existing surface
Keyboard Command: set_display_exist_ctr

Existing Surface

This command allows you to display the surface triangulation for the existing drawing.

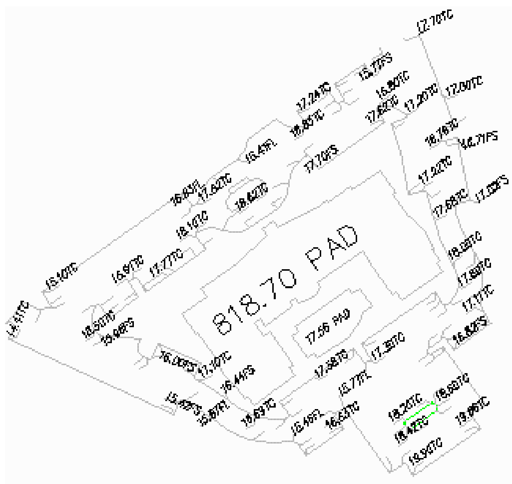


When Existing Surface is checked than all the triangulation for the existing will be displayed. If it is not checked, they will not be displayed.

Keyboard Command: set_display_exist_grd
Prerequisite: an existing surface

Design Drawing

This command allows you to display all the entitles on the layers that are grouped as part of the Design Drawing.



Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Design Drawing is checked than the design drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Design drawing in isolation or in accordance to the other drawings.

Keyboard Command: set_display_final_dwg
Prerequisite: Define Layer Target/Material/Subgrade

Design Contours

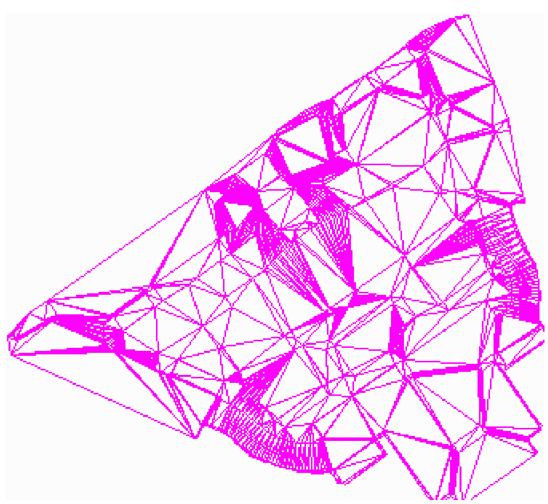
This command displays all the contours that represents the design surface (For contouring options see Display Options).Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

When Design Contours is checked than all the contours for the design will be displayed. If it is not checked they will not be displayed.

Keyboard Command: set_display_final_ctr
Prerequisite:design surface

Design Surface

This command allows you to display the surface triangulation for the design drawing.



When Design Surface is checked than all the triangulation for the design will be displayed. If it is not checked, they will not be displayed.

Keyboard Command: set_display_final_grd
Prerequisite: a design surface

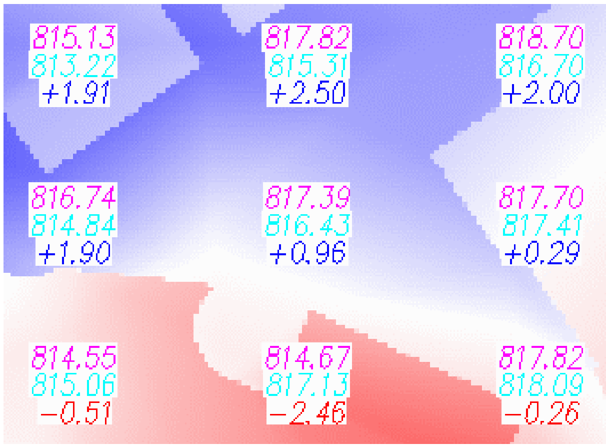
Cut/Fill Contours

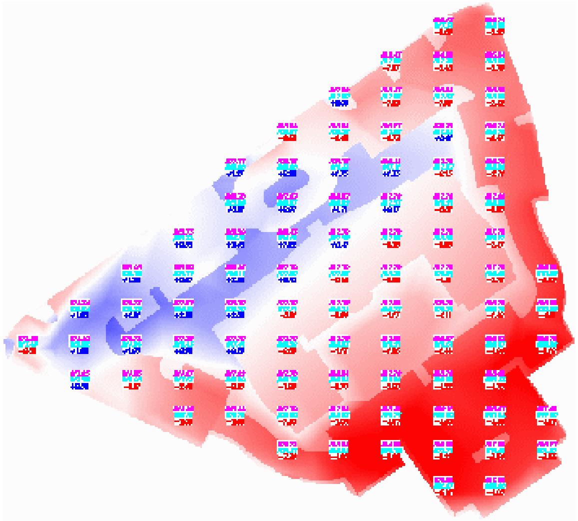
This command compares the existing and design surfaces and shows the cut/fill contours in blue for fill and red for cut. There is a Draw Only Cut/Fill Daylight option and Draw Labels option as part of the Display Options command (See Display Options for more information). Clicking on Cut/Fill Contours from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

Keyboard Command: set_display_cf_ctr
Prerequisite: elevation differences between existing and design

Cut/Fill Labels

This command displays the design elevation, the existing elevation, and the amount to either cut or fill right on the screen (See Display Options for information about labeling options). Picking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

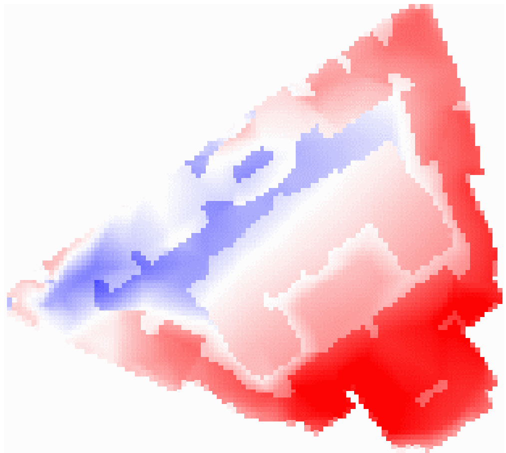




Keyboard Command: set_display_cf_txt
Prerequisite: existing and design surfaces

Cut/Fill Color Map

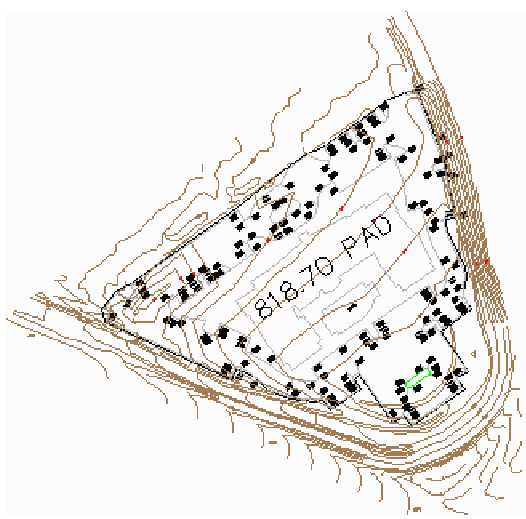
This command compares the existing and design surfaces and shows the cut/fill regions in blue for fill and red for cut (See Display Options for information on pixel resolution). Clicking on Cut/Fill Color Map from the menu runs the command and puts a check mark on the menu. Picking again turns it off.



Keyboard Command: set_display_cf_map
Prerequisite: existing and design surfaces

Other Drawing

This command allows you to display all the entities on the layers that are grouped as part of the Other drawing.



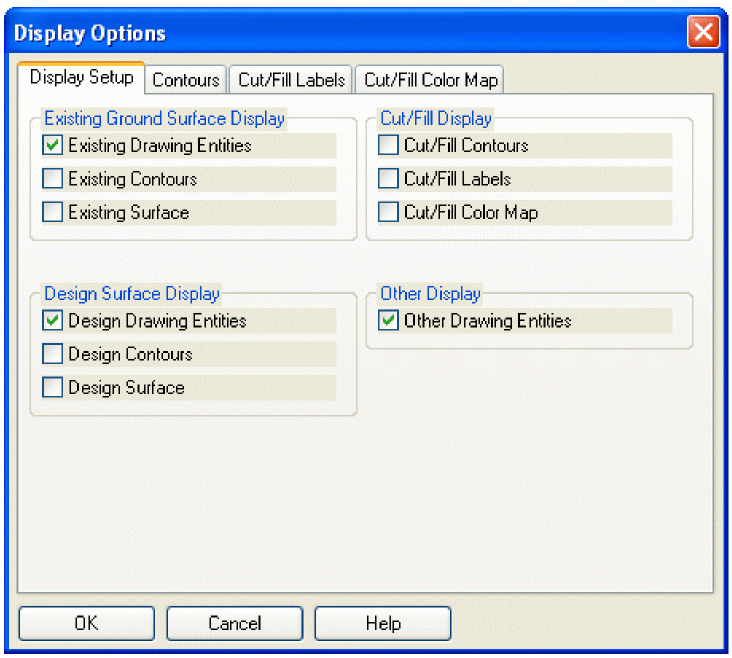
Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. Typically, most layers are listed under Other before they are assigned to Existing or Design. Some layers, like perimeter, are neither apart of the Existing or the Design drawing so they remain under Other. When Other Drawing is checked than the entities grouped under Other will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Other surface in isolation or in

accordance to the other surfaces. In this example, Existing, Design, and Other are all shown.

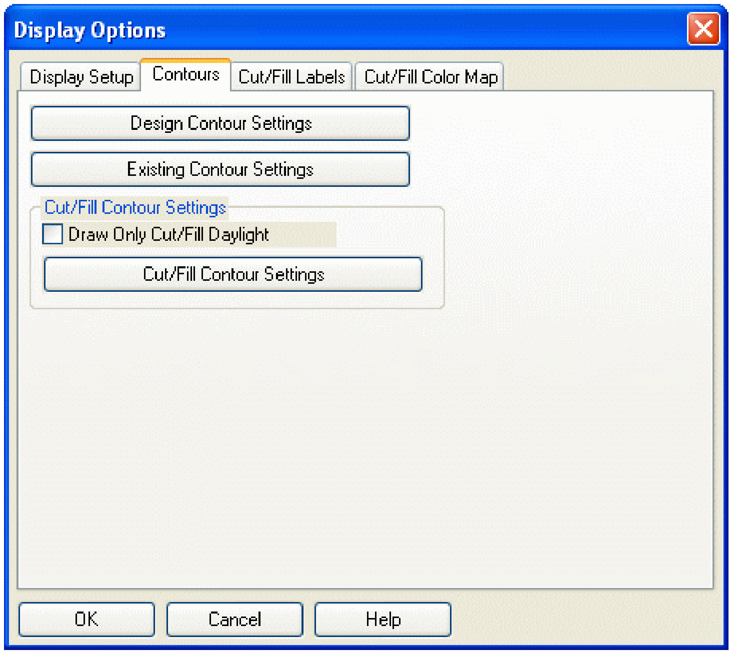
Keyboard Command: set_display_other_dwg
Prerequisite: Define Layer Target/Material/Subgrade

Display Options

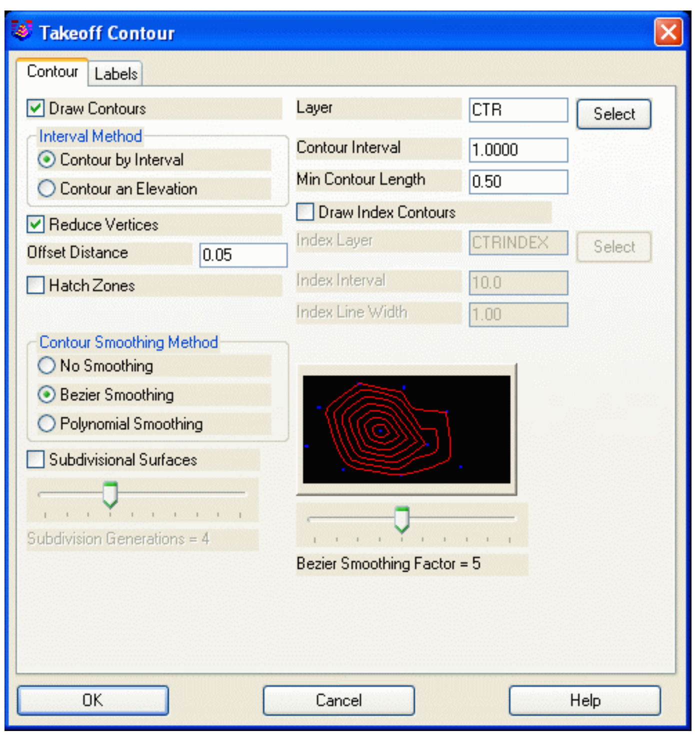
This command allows you to change the features of the different display commands. Note: You can toggle on/off the Existing, Design, and Other surfaces by right clicking with your mouse. To activate this feature type in "shortcutmenu" in the command line and then <1>. To turn off the feature type in <0>.



Display Setup: Here is the master list for the major things you can display, including: the Entities, Contours, and Surface for both the Existing and Design, Cut/Fill Displays, and Other Drawing Entities.



Contour Options: Here you can set the interval, the elevation difference between each contour, for the Existing, Design and Cut/Fill by clicking on their Contour Settings. You can also choose to draw only the daylight line between Existing and Design instead of the Cut/Fill contours at an interval.



Draw Contours

When this box is checked, the program will draw contour lines after triangulating. Otherwise, only the designated triangulation operations are performed. Specify the layer for contours in the edit box to the right.

Contour by Interval or **Contour an Elevation**
Select whether to contour by interval (ie: every 10 feet) or to contour a certain elevation. The elevation option allows you to contour specific values. For example, if you want just the 100ft contour, then select elevation and enter 100. The default mode is by interval.

Contour Interval

Specify the interval to contour. Note: If the above option is set to Contour an Elevation, then this field is used to specify the elevation to contour.

Minimum Contour Length

Contour lines whose total length is less than this value will not be drawn.

Reduce Vertices

This option attempts to remove extra vertices from the contour polylines which has the advantages of a faster drawing and smaller drawing size. Default is ON

Offset Distance

When the Reduce Verticies option is enabled, This value is the maximum tolerance for shifting the original contour line in order to reduce vertices. The reduced contour polyline will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour line to shift more from the original.

Hatch Zones

When activated, this option will allow you to hatch the area between the contours sequentially. A secondary dialog will load allowing the user to specify the hatch type and color.

Draw Index Contours

This option creates highlighted contours at a specified interval. When enabled, the fields for Index Layer, Index Interval and Index Line Width are activated.

Contour Smoothing Method

Select the type of contour smoothing to be performed. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces. The smoothing factor described below affects the smoothing bulge.

Bezier Smoothing Factor

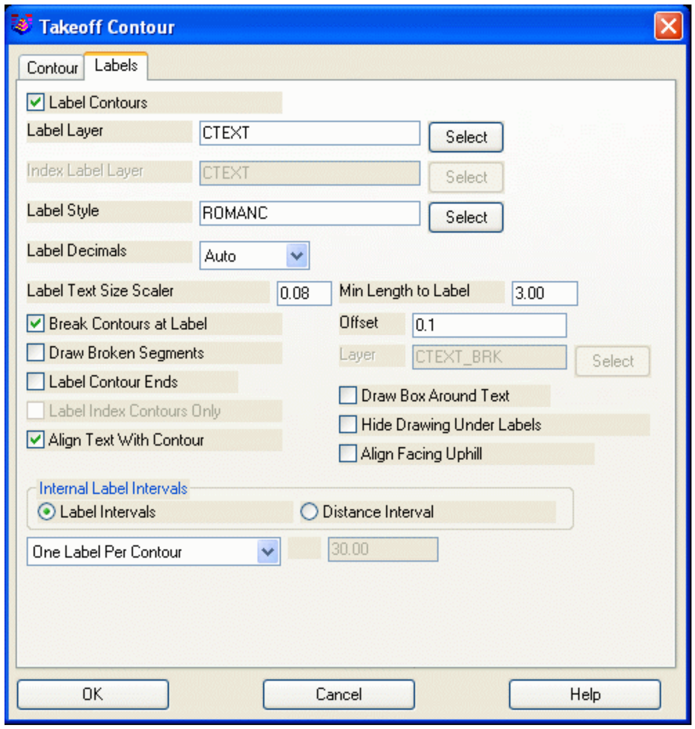
The contour preview window shows you an example of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect.

Subdivisional Surfaces / Subdivisions Generation

This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation, with the new temporary vertices raised or lowered to provide smoother contours. More generations increases the smoothness of the algorithm at a cost of increased processing time. If

Straight Lines are chosen as the contouring drawing method, then the contours are guaranteed never to cross. The original points of the surface model are always preserved. These modifications to the surface model are only for contouring purposes and are not written to the triangulation (.FLT) file or inserted into the drawing. If some contour movement is too small for appearance's sake, consider enabling Reduce Vertices.

Label Tab



Label Contours

When activated, contours will be labeled based on the settings below.

Label Layer

Specifies layer name for intermediate contour labels.

Index Label Layer

Specifies layer name for index contour labels.

Label Style

Specifies the text style that will be used for the contour label text.

Label Text Size Scaler

Specifies the size of the contour labels based on a multiplier of the horizontal scale.

Min Length to Label

Contours whose length is less than this value will not be labeled.

Break Contours at Label

When checked, contour lines will be broken and trimmed at the label location for label visibility. When enabled, the Offset box to the right activates. The Offset determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

Draw Broken Segments

When checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label Contour Ends

When checked, contour ends will be labeled.

Draw Box Around Text

When checked, a rectangle will be drawn around contour elevation labels.

Label Index Contours Only

When checked, only the index contours will be labeled. This option is active only when "Draw Index Contours" has been selected in the Contour tab of the main dialog.

Hide Drawing Under Labels

This option activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour is fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, and the contour lines themselves remain contiguous. This feature will also hide other entities that area in the immediate vicinity of the contour label.

Align Text with Contour

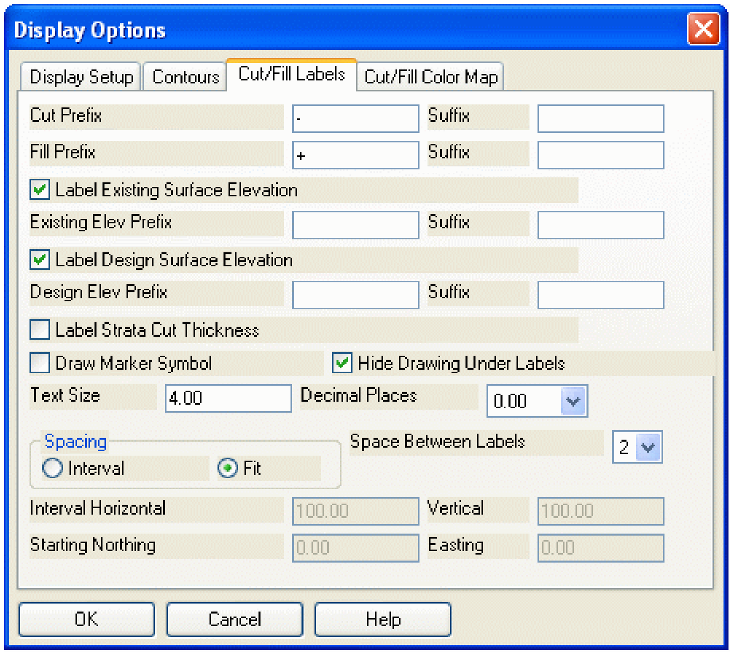
When checked, contour elevation labels will be rotated to align with their respective contour lines. This option also activates the Align Facing Uphill feature explained below.

Align Facing Uphill

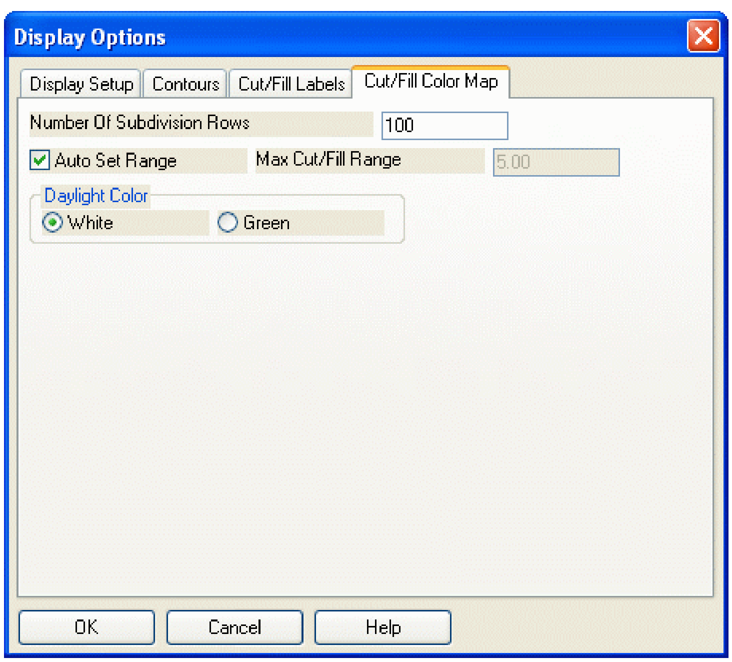
When checked, contour elevation labels will still be rotated to align with their respective contour lines, but the labels will be flipped in such a manner that the bottom of the text label will always be toward the downhill side of the contours. So as the labels are read right side up, you are always facing uphill.

Internal Label Intervals

Choose between label intervals or distance interval. Label intervals will label each contour with a set number of labels. Distance interval lets you specify a distance between labels.



Cut/Fill Label Options: Here you can customize the Cut/Fill labels. Text can be added either before or after the Cut/Fill amount, the Existing elevation, and the Design elevation with the Prefix and Suffix fields. You can also choose whether or not to display the Existing Surface elevations, the Design Surface elevations and Strata Cut Thickness. Carlson TakeOff gives you the option to draw a marker symbol for where each label represents. You can also hide the drawing under the labels so that you can read the labels clearly. Text Size chooses the text size for each line of the label. Decimal Places sets to how many decimal places the labels will report. The Spacing of the labels can be determined by intervals or by a selected number of spaces. The size of each space is determined by the Text Size.



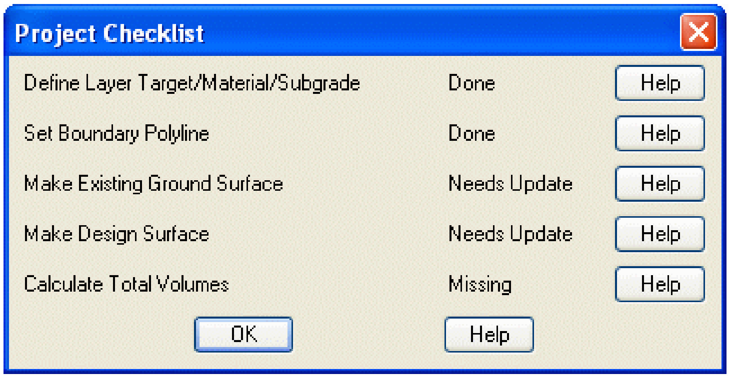
Cut/Fill Color Map Options: Number Of Subdivision Rows is the number of blocks both horizontally and vertically in the Color Map. If the box reads 100 that means 100 blocks left to right and 100 blocks up and down or 10,000 total pixels. A higher the Number Of Subdivision Rows will make the Color Map sharper, however too high number can cause Carlson TakeOff to run slower. Auto Set Range will automatically set the red to blue scale for your cut/fill levels. However, if you desire greater contrast, then use Max Cut/Fill Range to manual set the range. Use lower numbers for greater contrast. The Daylight Color can be set to be either White or Green.

Keyboard Command: tk_display_options
Prerequisite: a drawing

Help Menu 17

Project Checklist

This command allows to check the status of steps needed to calculate total volumes.



Prerequisite: none
Keyboard Command: tk_checklist

On-Line Help

This command opens the Carlson TakeOff on-line Help File.

Prerequisite: None

Keyboard Command: [F1] or HELP

Training Movies

This command opens an application that lets you choose from several training movies. The movies provide instruction for all aspects of Carlson TakeOff.

Prerequisite: None

Keyboard Command: RUN_MOVIES

Carlson WebSite

This command brings you to the Carlson Home webpage.

Prerequisite: an internet connection
Keyboard Command: ._browser Enter Web location (URL) <http://www.carlsonsw.com>:
"http://www.carlsonsw.com/"

About Carlson Survey

Displays the Carlson TakeOff version number, serial number, license information, and copyright information. You can run the registration wizard by clicking the Change Registration button on this dialog.

Prerequisite: None

Keyboard Command: ABOUT_SCAD

Tutorials Menu 18

Takeoff Tutorial: CAD File Takeoff From Start To Finish

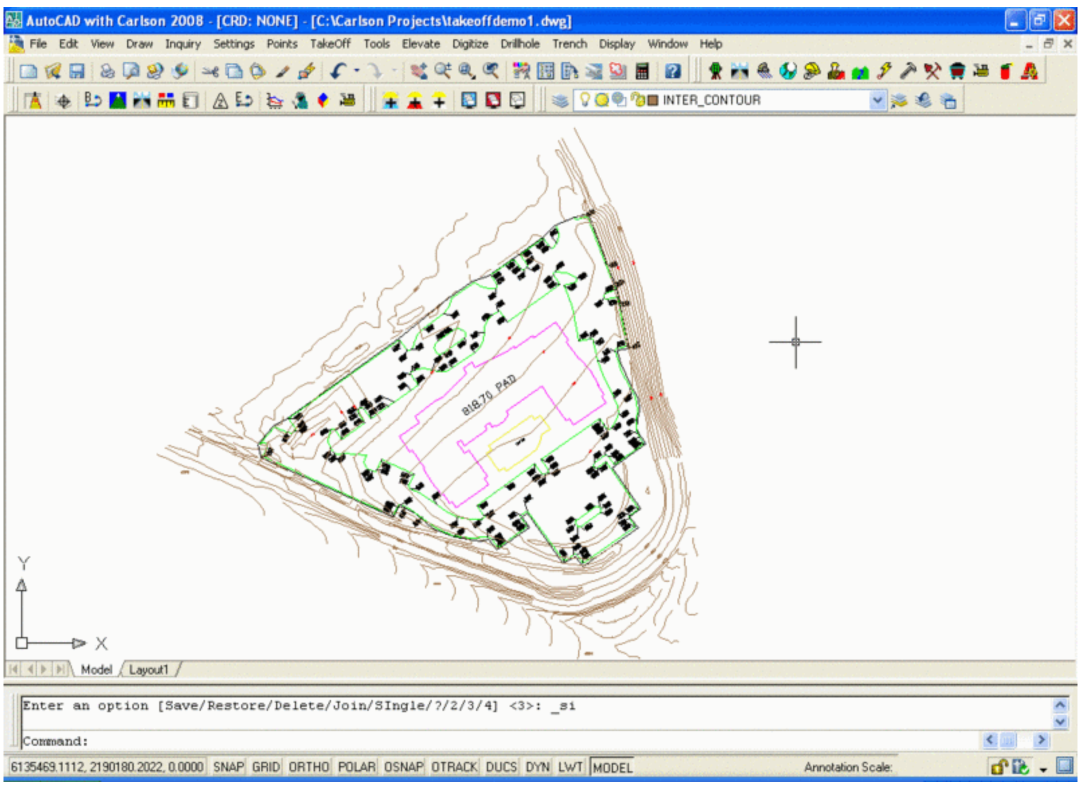
Note: Completing these tutorials will alter the drawing files (demo1.dwg, demo2.dwg, demo3.dwg). If you would like to run through the tutorials a second time, copy over the original drawing files (.dwg) into a new folder under C:\Carlson Projects.

Step 1 (Start Takeoff):

Click the icon for Takeoff on your desktop or from the toolbar to launch the program. You may be presented with a "Startup Wizard" dialog and if so, click Exit.

Step 2 (Open Drawing):

From the File menu, choose Open and select demo1.dwg from the Carlson Projects folder.



Now we can begin to process this drawing. The main Takeoff commands are listed in processing sequence in the Takeoff pull-down menu. Many of these commands are also grouped as icons in the toolbar shown here.

Step 3 (Drawing Cleanup):

From the Takeoff pull-down menu, choose Drawing Cleanup. Typically, drawings have lots of drafting fixes that must be done before the surfaces can be modeled. This command will apply the selected cleanup functions on the drawing to help automate the cleanup. Here's a brief description of the most important of these functions:

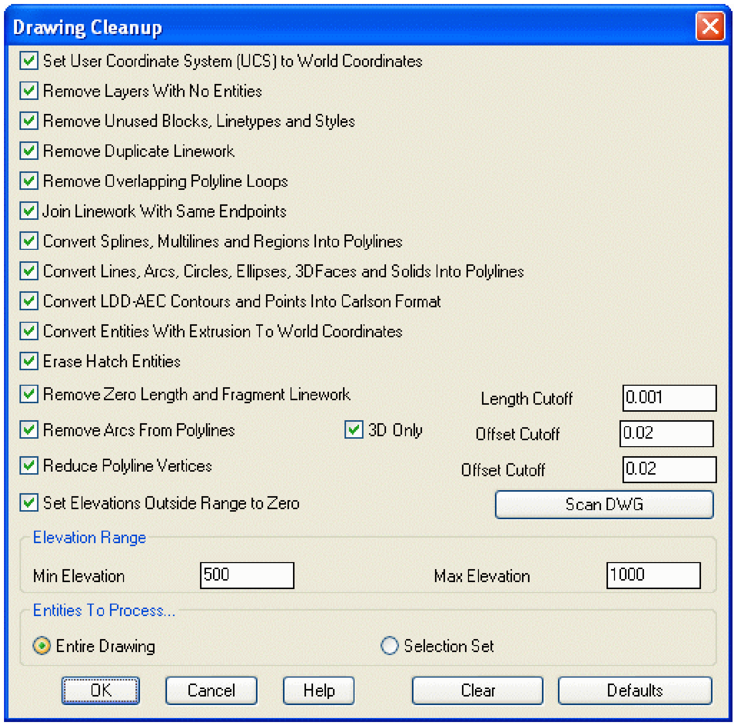
Remove Layers With No Entities: Drawings often have lots of layers. This routine removes layers that have no entities in the drawing so that we don't have to deal with them.

Join Linework With Same Endpoints: This routine will take linework that is broken into multiple segments and join them into a single linework entity. For example, it will join together broken segments of a contour polyline into a single polyline.

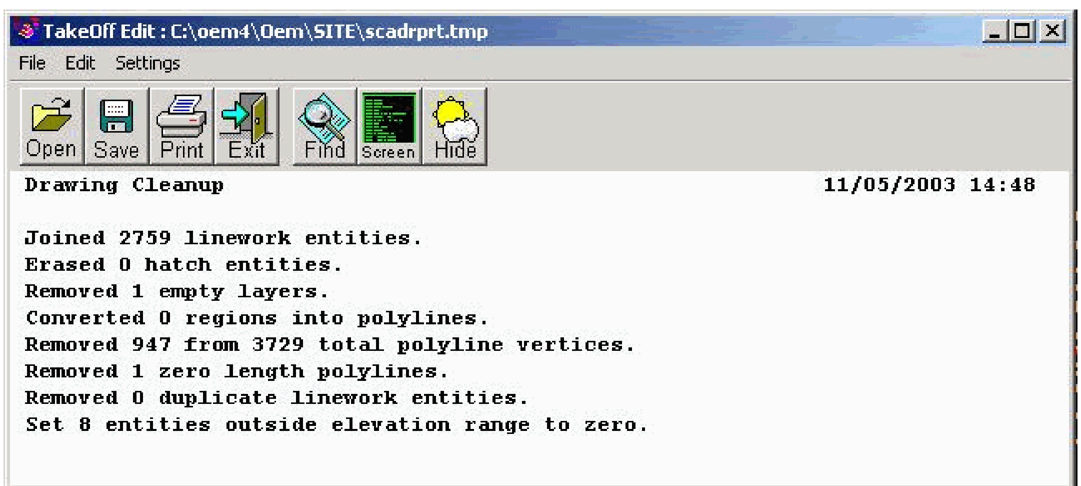
Reduce Polyline Vertices: This routine removes extra vertices from polylines as long as the removing does not shift the polyline more than the specified Offset Cutoff. This will reduce the size and complexity of the drawing.

Set Elevation Outside Range To Zero: In case the drawing contains entities that are outside the range of valid elevations for the site, this routine will set them to zero elevation. The program treats zero elevations as "no elevation" and modeling will filter out these zero elevation entities.

For this site, the elevations are around 800. So let's set the Min elevation to 500 and the Max elevation to 1000. The cleanup will set any entities outside this elevation range to zero. With other Takeoff functions, we can later assign proper elevations to any of these zero elevation entities that need to be used in modeling.



Once the Drawing Cleanup options are set as shown, pick OK. When the cleanup is done, the program will show a report of the cleanup results. Pick the Exit button to exit the report viewer.

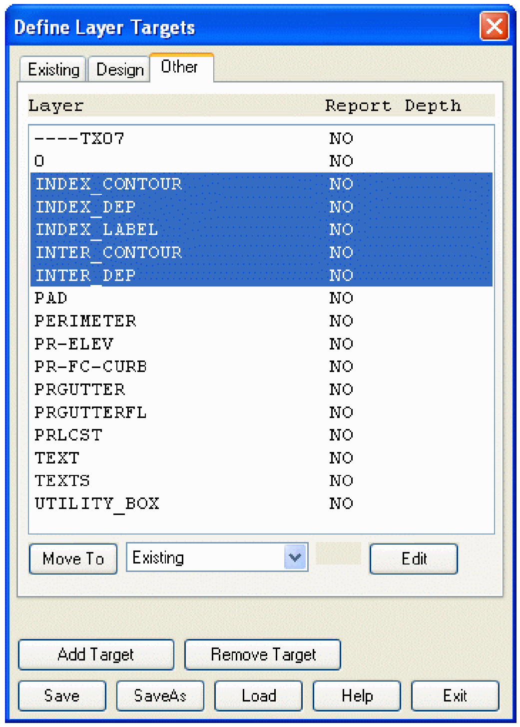


Step 4 (Layer Targets):

From the Takeoff menu, choose Define Layer Target/Material/Subgrade. Every entity (line, polyline, point, etc) in the drawing is assigned a layer name. Takeoff uses the entity layer names to define which entities are for the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces. The drawing entities are assigned their target surface by their layer name. For example, if polylines representing design contours are on the layer "Final", then "Final" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other.

The Define Layer Targets dialog has three lists of layers: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog.

In this drawing, all the contours are for the existing ground surface. In the layer list, all the layers that start with INDEX and INTER are for these contours. So highlight these layers and then choose Move To Existing. To highlight multiple layers at a time, hold down the keyboard Ctrl key while picking with the mouse.



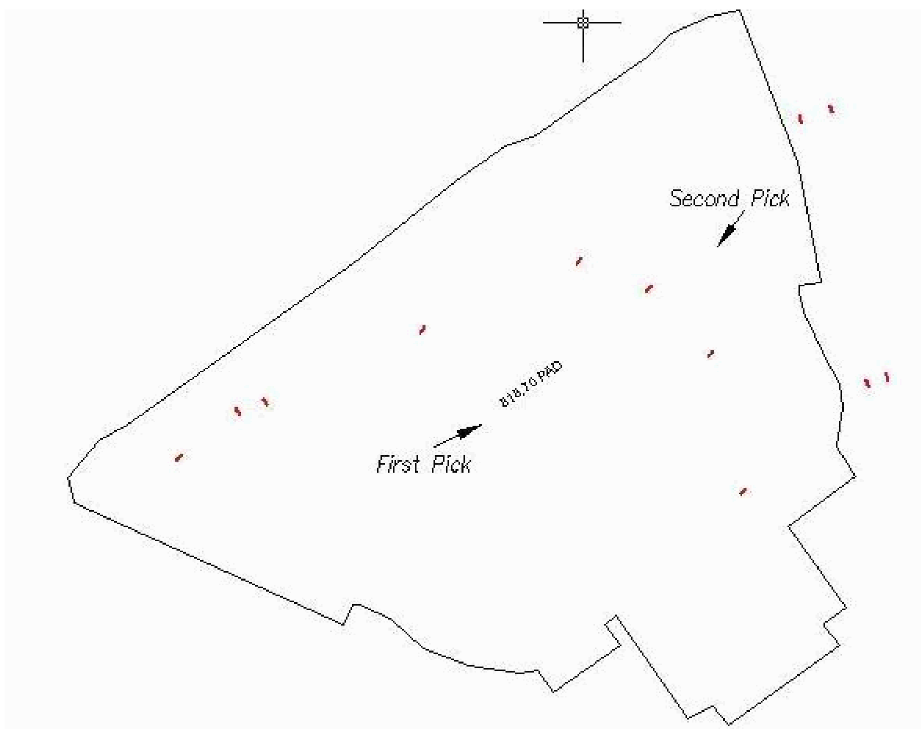
Next move the layer names that start with "PR" (for proposed) to the Design surface by highlighting these layers and choosing Move To Design. Also move the layer "PAD" to design.

Next pick the Save button to save our changes and then pick Exit.

There are more tools for assigning layer targets. In the Display menu, you can turn on/off whether to display layer targets by using Existing Drawing, Design Drawing and Other Drawing. For example, when Design Drawing is checked, then picking this menu item will uncheck it and turn off all the layers for the design surface. Likewise, picking Design Drawing when it is unchecked will make it checked and turn on the design surface layers.

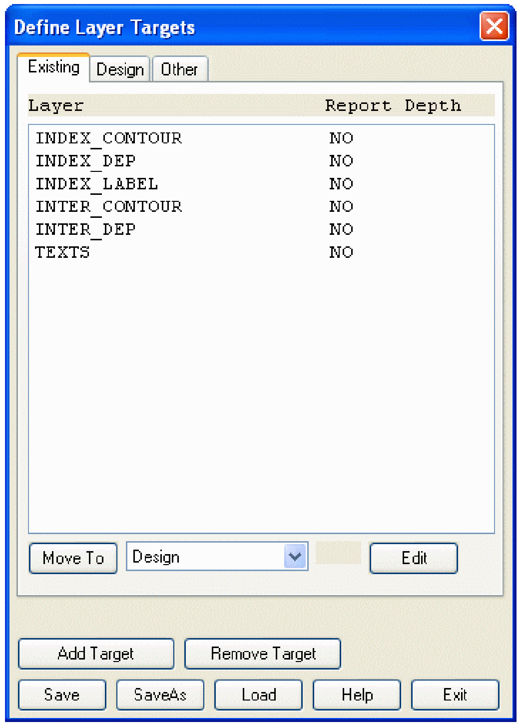
Practice turning on/off the Existing, Design and Other Drawing in the Display menu. When only Existing Drawing is on, you should see just the contours. When only Design Drawing is on, you should see just the design polylines and leader labels. When only Other Drawing is on, you should see the entities that are assigned to neither existing nor design.

Some of these layers we do want to assign to existing and design. To better see the entities, zoom in on them using the View->Zoom->Window command and pick two points that make a window around the entities as shown. Once zoomed in, you can see a text label of "818.70 PAD" which is for the design surface. Labels "817.00", "818.00" and "819.00" are contour labels for the existing contours.

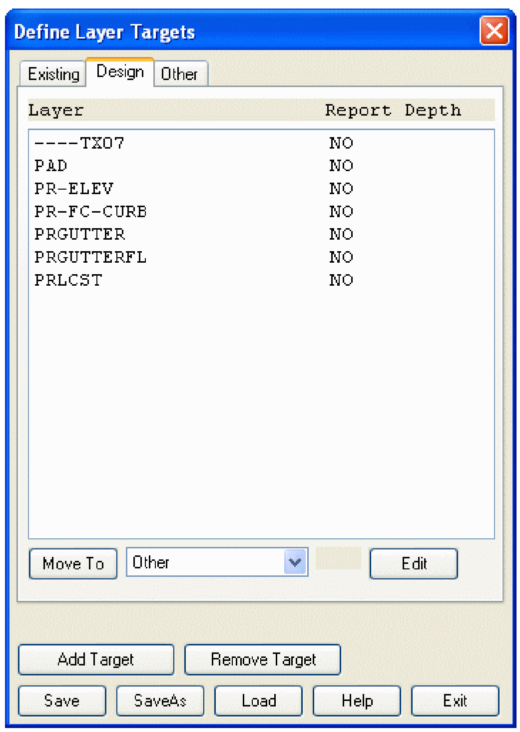


There are a few commands in the Inquiry menu to find out the layer names for these entities: List, Layer ID and Drawing Inspector. Let's run the Layer ID command and pick the "818.70 PAD" label. At the Command line, it reports this layer is "—-TX07". Next pick the "818.00" label and it reports this layer is "TEXTS". Now go up to the Takeoff Menu->Set Layer For Design and pick on label for the Building Pad. You will notice that if Design Drawing was checked off under Display, the Pad label will turn off. This is because the layer "—-TX07" is now apart of the Design Drawing. Go to the Takeoff Menu again and this time pick Set Layer For Existing and select the "TEXTS" layer. Press Enter, this will put the "TEXTS" layer on Existing.

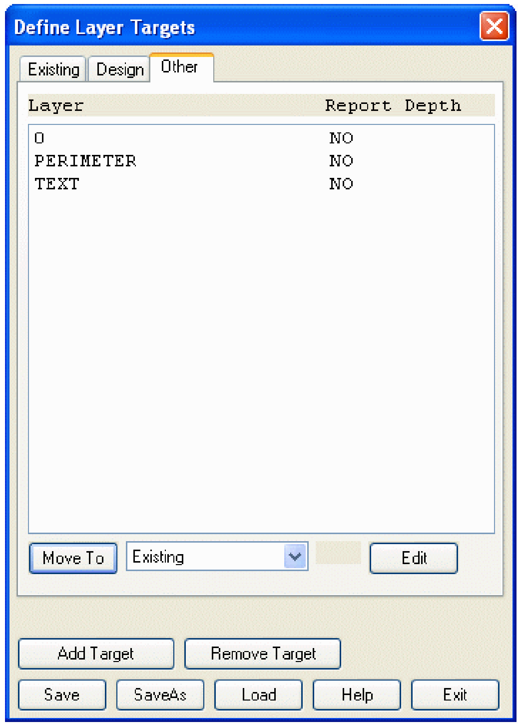
Next run Define Layer/Material/Subgrade and you will notice that "—-TX07" is now under Design and "TEXTS" is under Existing. Check that your Layer Targets match the three lists shown below. Then pick Save and Exit.



Define Layer Targets
Existing
Design
Other
Layer
Report Depth
INDEX_CONTOUR NO
INDEX_DEP NO
INDEX_LABEL NO
INTER_CONTOUR NO
INTER_DEP NO
TEXTS NO
Move To
Design
Edit
Add Target
Remove Target
Save
SaveAs
Load
Help
Exit



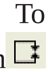
Define Layer Targets
Existing
Design
Other
Layer
Report Depth
----TX07 NO
PAD NO
PR-ELEV NO
PR-FC-CURB NO
PRGUTTER NO
PRGUTTERFL NO
PRLCST NO
Move To
Other
Edit
Add Target
Remove Target
Save
SaveAs
Load
Help
Exit



Step 5 (Define Material/SubGrade):

Besides assigning target surfaces by layer, layers are also used to define material names and subgrades depths. By assigning material names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface. The polylines used for subgrade depth must be closed polylines. Takeoff supports nested subgrade polylines for exclusion areas such as islands by counting how many subgrade polylines surround an area. If the number is odd, then the area is inside the subgrade. Otherwise the area is not part of the subgrade.

<>First, we need to know the layer names for our subgrades. Go to the Display menu and check on Design Drawing, uncheck Existing Drawing and uncheck Other Drawing. Then run Inquiry->Layer ID and pick the large pad polyline. It reports that this layer is PAD. Next use Layer ID to pick the curb polyline. It reports that this layer is PR-FC-CURB.

Next we need to make sure that these polylines are closed. In this example, the outside curb polyline is open at the top. To close the polylines, run Edit->Polyline Utilities->Edit Polyline->Close Polylines or pick on the Toolbar icon  . Then pick each of the pad and curb polylines and press Enter when done selecting. Here are the Command line prompts:

Select Polylines to set closed.

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: 1 found, 4 total

Select objects: 1 found, 5 total

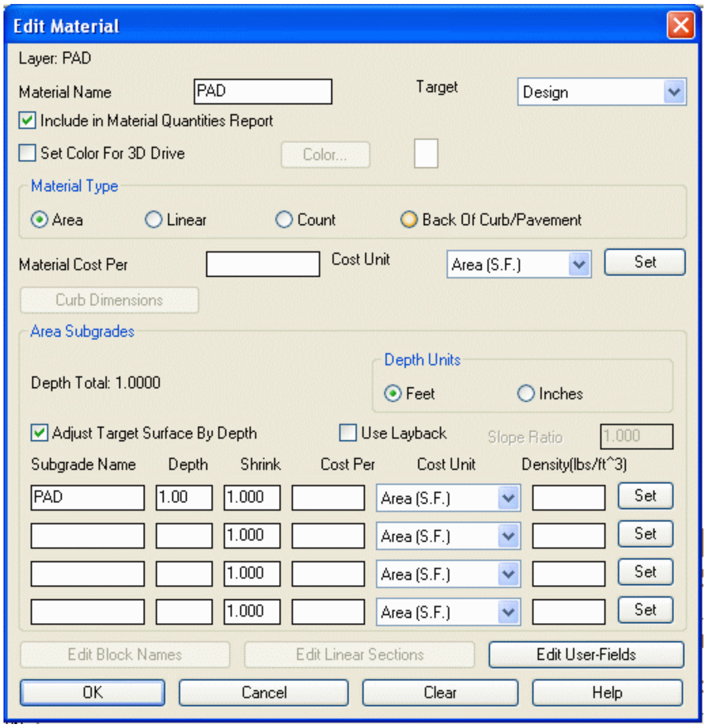
Select objects: 1 found, 6 total

Select objects: (Press Enter)

5 polylines already closed.

Closed 1 polylines.

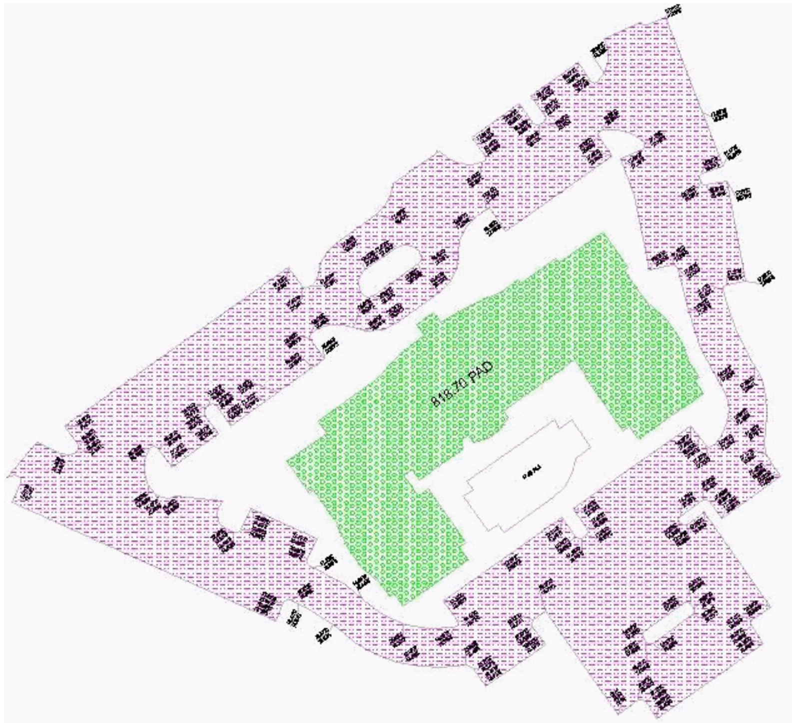
Now run Define Layer Target/Material/Subgrade and pick the Design tab. Highlight layer PAD and pick the Edit button. A dialog appears for defining the pad material properties. Check on the Include In Material Report option, enter the Material Name as "Pad", set the first subgrade name to "Pad", and set the Depth as 1. Once the dialog is filled out as shown, pick OK.



Next pick layer PR-FC-CURB and choose Edit. In the Edit Materials dialog, check on Include In Material Report, set the Material Name to "Pavement", set the first subgrade name to "Pavement", and set the Depth to 1.5. Then pick OK.

<>To save the subgrade changes, pick the Save button on the Define Layer Targets dialog. Then choose Exit.

Now let's visually verify the subgrade areas. In the Takeoff menu, run Subgrade Areas->Hatch Subgrade Areas. There is a dialog to select which subgrade to hatch. Choose the Pavement. Then there is a dialog for the hatch pattern and color. Click OK. Then run Hatch Subgrade Areas again. This time choose Pad and set the hatch pattern to Hex with green color. The resulting hatch areas show where the subgrade is applied. Notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Also note that the smaller pad area is not hatched because this polyline layer is different than the bigger pad polyline. When finished viewing the subgrade areas, run Takeoff->Subgrade Areas->Erase Subgrade Hatches.



Step 6 (Elevate Drawing - 2D to 3D):

Takeoff will model the existing ground and design surfaces based on points, lines and polylines with elevation. It is essential for these drawing entities to have correct elevations in order to get correct surface models. Often the provided drawings will have the drawing entities at elevation of zero with text labels indicating the true elevation. Takeoff has many tools for assigning elevations to these entities.

<>To help visualize which entities need to be assigned elevation, Takeoff will color entities at zero elevation in grey. As entities get assigned elevation, they return to their original color. This elevation coloring is applied to layers that have been assigned to the existing or design surfaces.

Let's start by working on the existing surface. To isolate the existing entities, go to the Display menu and check on Existing Drawing, uncheck Design Drawing and uncheck Other Drawing. In the Inquiry menu, there are commands for checking elevations. To check the elevation of the contour polylines, run the Inquiry->List Elevation and pick a contour polyline At the command line, it reports the elevation.

Select Entity: pick pad polyline

Elevation: 816.0000

Select Entity (Enter to end): press Enter

In this example, the existing ground surface is defined by just contour polylines and these polylines already have elevation. So there are no changes needed for preparing the existing surface entities. If the contour polylines were at zero elevation, then you could use the Elevate->Assign Contour Elevation commands.

Next let's prepare the design surface. To isolate the design entities, go to the Display menu and check on Design Drawing, uncheck Existing Drawing and uncheck Other Drawing. Notice that all the design linework is greyed because it is at zero elevation. Run the List Elevation command and pick the main pad polyline. At the command line, it confirms that the elevation is 0.

To set the pad polyline elevation, run Elevate->Set Polyline To Elevation. Enter an elevation of 818.7 (based on the text label). At the Select objects prompt, pick the bigger pad polyline and press Enter.

New Elevation <0.0000>: 818.7

Select Lines, Arcs, Circles or Polylines for elevation change.

Select objects: (pick the pad polyline) 1 found

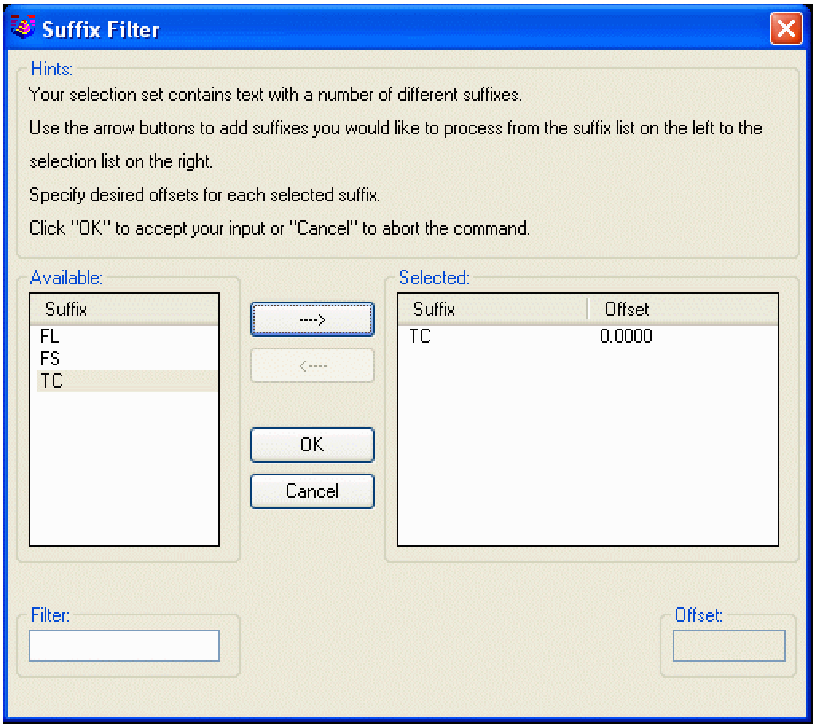
Select objects: press Enter

LWPOLYLINE

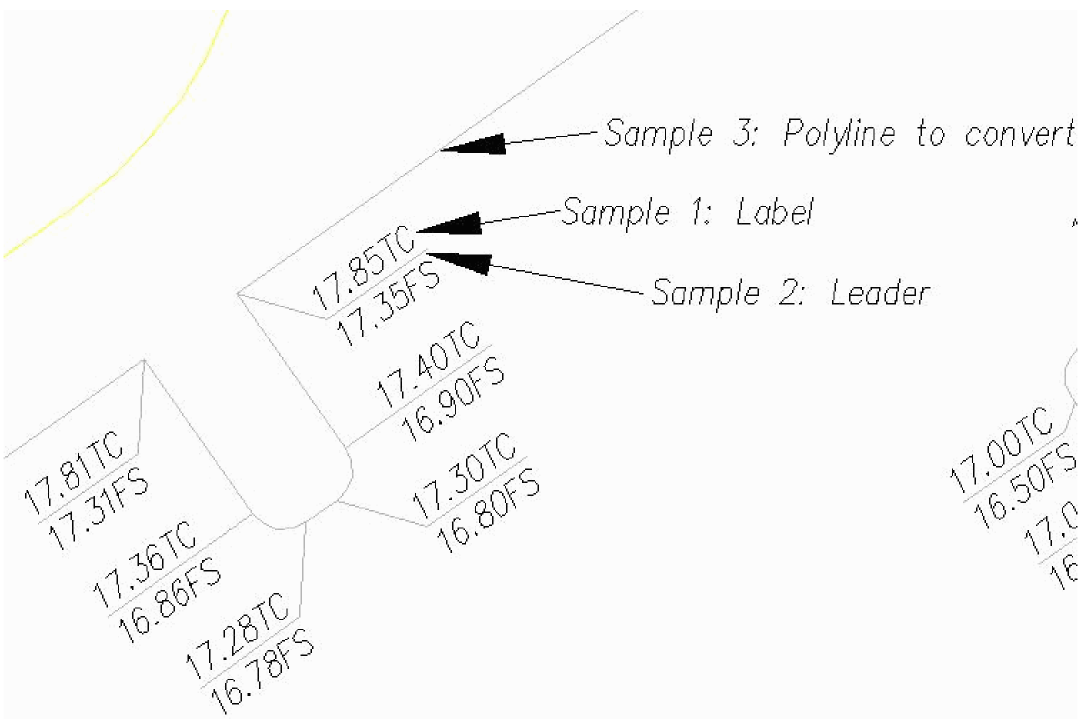
Number of entities changed> 1

Next let's set the elevation of the smaller pad under the main pad. First, use View->Zoom->Window to zoom in around the smaller pad so that we can read the text label. The label of "17.56" is short for 817.56. In this example, the 800 was dropped from many of the elevation labels to save on label clutter. Run Set Polyline To Elevation again. This time enter an elevation of 817.56 and pick the smaller pad polyline. Then run View->Zoom->Previous to get back to the full view of the site.

Finally, we need to set the elevations for the curb polylines. First, use View->Zoom->Window to zoom in around some of the curb labels below the smaller pad. Then run Elevate->2D to 3D Polyline->Text With Leader. This command will assign the elevations from the labels to the polylines by following the label leader to find the position on the polyline. For polyline vertices without elevation labels, the elevations will be interpolated from the other labels. Before processing, this routine prompts for samples of the elevation label, the leader and the polylines to convert. Then you can select all the entities in the drawing and the routine will sort the labels, leaders and polyline by the sample layers and assign the elevations. For this example, pick one of the labels with a "TC" suffix as the elevation text sample. Then pick the leader line for the annotation leader sample. Then pick the curb polyline for the polyline to convert sample. At the Select objects prompt for processing, type "all" to select all the drawing entities and press Enter. For the elevation to add, enter 800 so that labels like "17.81" get assigned as 817.81.



Next a dialog appears for selecting which labels names to use. When Takeoff detects different text labels within the elevation labels, you need to choose which ones to process. In this case, we only want the labels with "TC". So highlight TC, pick Add and then pick OK.



Select sample of elevation text: pick label

Select sample of an annotation leader: pick leader line

Select sample of a polyline to convert: pick curb polyline

Select polylines to convert, leaders and elevation labels to process.

Select objects: all

Select objects: press Enter

Joining adjacent polylines...

Reading the selection set ...

Enter elevation to add to label values <0.00>: 800

Pre-processing entity #420 of 420

Processing leader #141

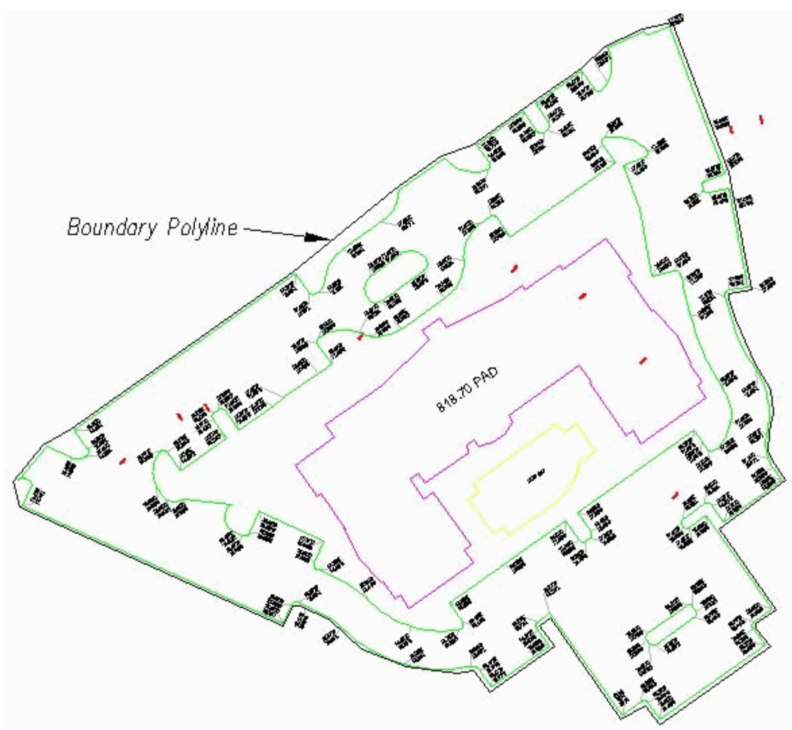
Remaking polyline #4

All the curb polylines now have elevations.

<>Now run View->Zoom->Previous to return to the full site view. The design polylines should now have colors because the elevations are assigned.

Step 7 (Boundary Polyline):

The limits of the site are defined by a closed polyline. This polyline is used as the boundary for the models and the volumes. In this example, there is a closed polyline on the PERIMETER layer. The layer target for this layer is Other. Go to the Display menu and check on Other Drawing so that the perimeter is displayed. Then run Takeoff->Boundary Polyline->Set Boundary Polyline and pick the perimeter polyline. This selected polyline is now set as the boundary polyline for the rest of the Takeoff routines.

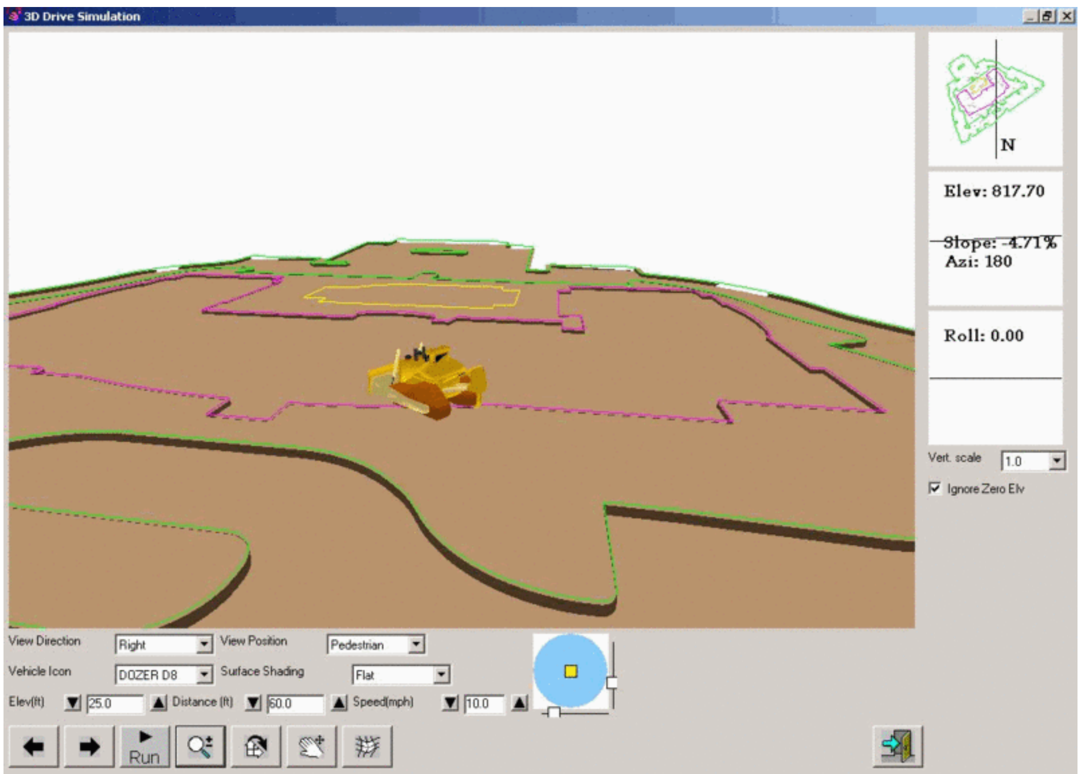


Step 8 (Model Existing and Design Surfaces):

To calculate volumes, Takeoff needs two surfaces: existing ground and design. These surfaces are modeled by triangulation. With the preparation of the previous steps, we're now ready to make the models. The drawing entities have been cleaned up, assigned elevations and assigned target surfaces by layer. Making the models is now a one step process. To make the existing ground surface, run Takeoff->Make Existing Ground Surface. The program will process the entities and make the triangulation surface. Then to make the design surface, run Takeoff->Make Design Surface.

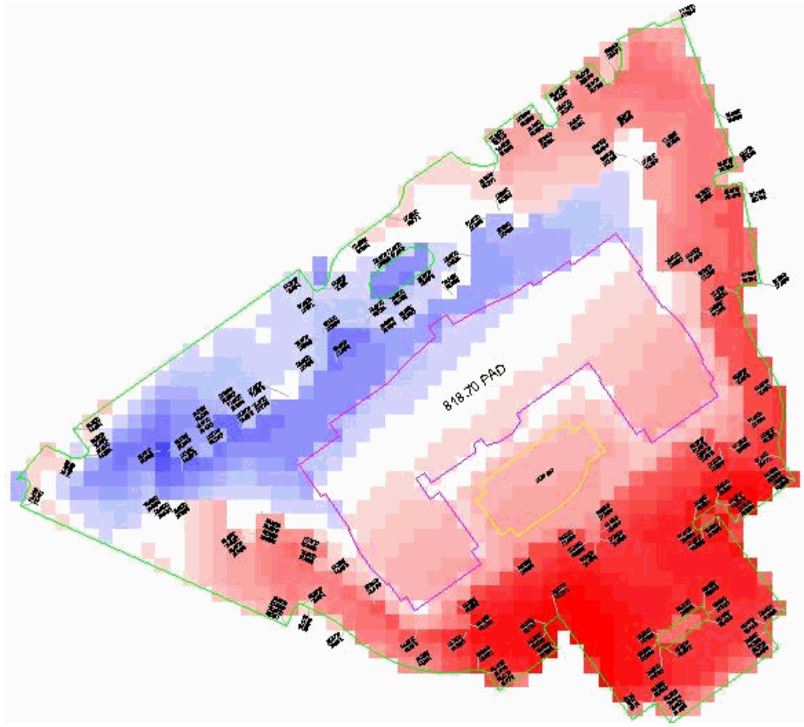
Step 9 (3D Drive Simulation):

As a visual check that the design surface modeled correctly, let's run the Tools->3D Drive Simulation command. This routine shows a 3D view of the site and allows you to drive around. This is a good way to check that the surface modeled correctly. We want to make sure that there are no elevation spikes and that the subgrade depths are modeled. To drive the site, choose a View Direction, View Position and Vehicle. Then pick the Run button and use the arrow keys to turn. Pick the Stop button to pause the moving. You can also try the Surface Shading options for different views of the surface. When done with the 3D Drive Simulation, pick the Exit button (Arrow with door image).



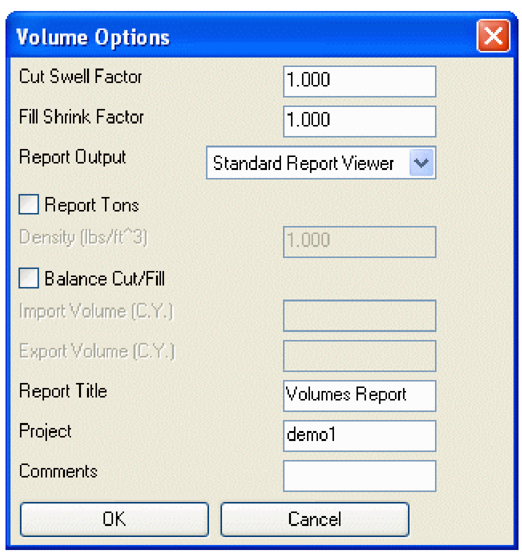
Step 10 (Cut/Fill Color Map):

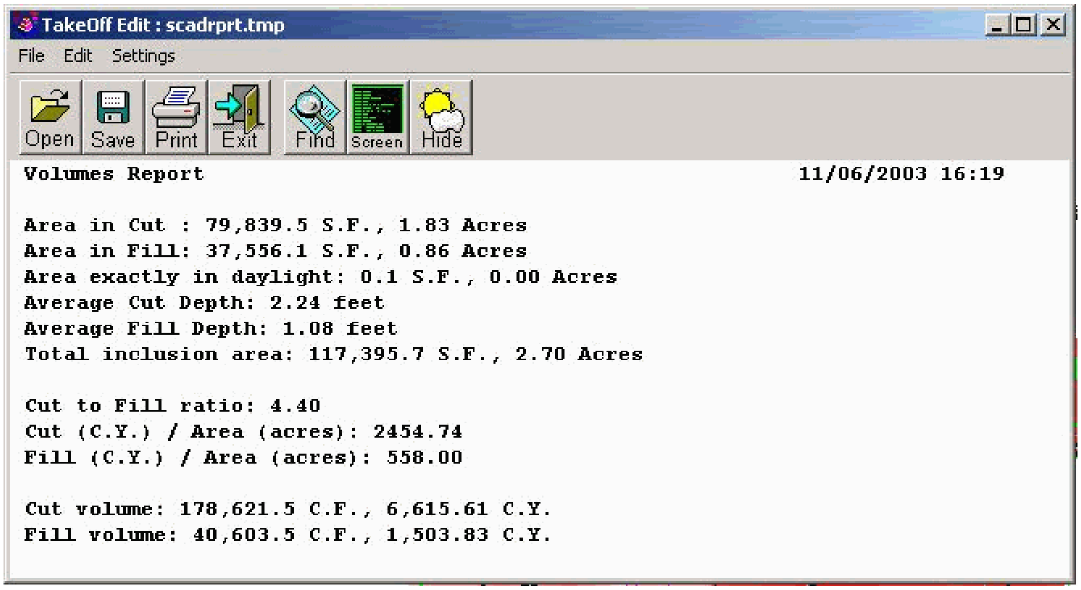
Cut/Fill color maps can be used for a visual output of the site cut/fill areas and also serves as another check that the models are correct. In the Display menu, choose Cut/Fill Color Map. Cut areas are drawn in different shades of red for different depths of cut while fill areas are drawn in blue. To change the resolution of the color blocks, run Display->Display Options and change the Cut/Fill Color Map Subdivisions. This parameter is the number of rows and columns of color blocks to create. To turn off the color map, go to the Display menu and pick Cut/Fill Color Map to uncheck it.



Step 11 (Calculate Volumes):

To calculate volumes, run the Takeoff->Calculate Total Volumes command. There is an options dialog for setting the cut swell factor and fill shrink factor. These values get multiplied into the cut/fill volumes. Set these factors as desired and click OK. Then the routine calculates the volumes and display the report which includes the cut/fill volumes and areas. The report can be printed or saved to a file. Pick the Exit button to exit the report viewer.





Volumes Report 11/06/2003 16:19

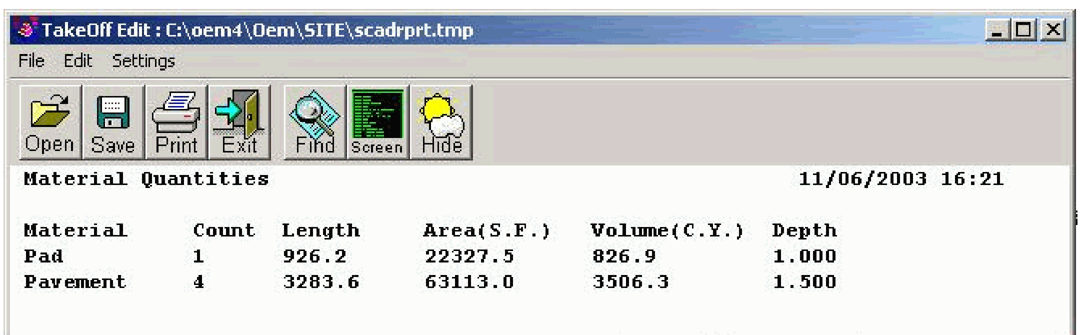
Area in Cut : 79,839.5 S.F., 1.83 Acres
Area in Fill: 37,556.1 S.F., 0.86 Acres
Area exactly in daylight: 0.1 S.F., 0.00 Acres
Average Cut Depth: 2.24 feet
Average Fill Depth: 1.08 feet
Total inclusion area: 117,395.7 S.F., 2.70 Acres

Cut to Fill ratio: 4.40
Cut (C.Y.) / Area (acres): 2454.74
Fill (C.Y.) / Area (acres): 558.00

Cut volume: 178,621.5 C.F., 6,615.61 C.Y.
Fill volume: 40,603.5 C.F., 1,503.83 C.Y.

Step 12 (Material Quantities):

To report the quantities, run the Takeoff->Material Quantities->Standard Report routine. The report includes the count, length, area and volume for each type of material that was assigned for reporting in the Define Layer Target/Material/Subgrade command. The Material Quantities->Custom Report routine can be used to reporting these values with control of the report format and the option to export to Excel.



Material Quantities 11/06/2003 16:21

Material	Count	Length	Area(S.F.)	Volume(C.Y.)	Depth
Pad	1	926.2	22327.5	826.9	1.000
Pavement	4	3283.6	63113.0	3506.3	1.500

Takeoff Tutorial: Road Design with Volumes

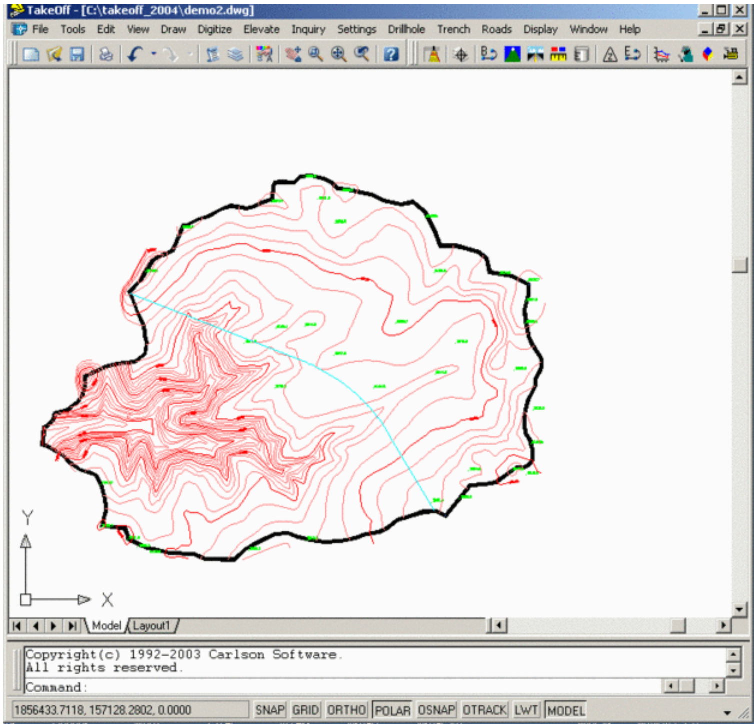
This lesson takes a drawing file through the steps of road design.

Step 1 (Start Takeoff):

Click the Windows icon for Takeoff to launch the program. You may be presented with a "Startup Wizard" dialog and if so, click Exit.

Step 2 (Open Drawing):

From the File menu, choose Open and select DEMO2.dwg from the Takeoff Work folder (ie. C:\Program_files\Carlson_Takeoff_2004\WORK\DEMO2.dwg).

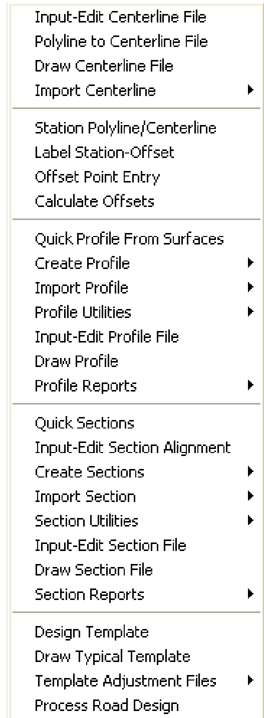


Step 3 (Existing Ground Surface):

For the road cut/fill slopes to be created, it needs the existing ground surface to tie into. First we need to define the layers of the existing ground surface. Run Takeoff-> Set Layer For Existing, pick on both a light-lined and heavy-lined contour, a spot elevation, and press Enter. This will set the layers CTR, CTRINDEX, and TO-PREM-SPOT-PNT to the existing surface. The limits of the site are defined by a closed polyline. Run Takeoff->Boundary Polyline->Set Boundary Polyline and pick the black polyline around the perimeter. This selected polyline is now set as the boundary polyline for the existing surface. To make the existing ground surface, run Takeoff->Make Existing Ground Surface.

Step 4 (Creating a Centerline):

Now we will use commands under the Roads menu.



Notice that the Roads menu is broken down into four sections: Centerline commands, Profile commands, Section commands, and Template commands. A road needs design input from all four sections to be created. It does not matter in what order the commands are run, but in this example we will run the commands in descending order.

A centerline file is necessary for the final road design routine. We will do the simplest variation, which is picking a polyline. There are other methods to design a centerline, and they are documented in the manual. Go to Polyline to Centerline File in the first grouping of commands under Roads. A file selection dialog will appear. Enter a centerline file name of demo2.cl and pick save. Follow the prompting:

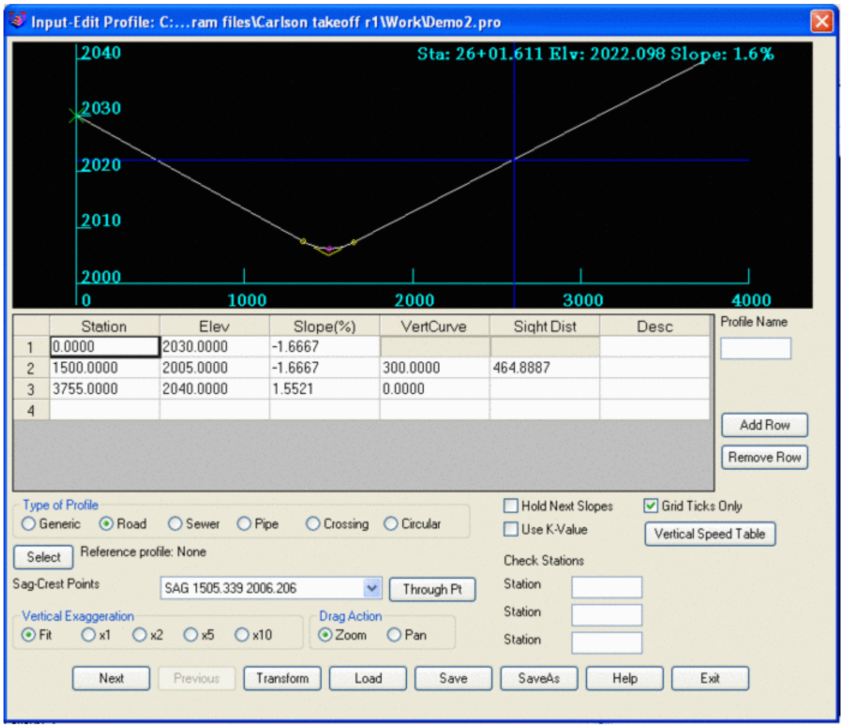
Beginning Station <0+00>: Press Enter
Polyline should have been drawn in direction of increasing stations.
Select polyline that represents centerline: Select the polyline that crosses the middle of the site with the layer name CLINE.
Station North(y) East(x) Description

0.0000 159718.2034 1857460.9166 LI
1798.2055 159058.6315 1859133.7908 PC
2912.2263 158347.2903 1859964.4134 LI
3755.6840 157619.5351 1860390.7855 LI
Press ENTER to continue.

Your Command Line should have the same values as these, as they are from the same line. Hit the F2 key or press Enter to return to the main screen.

Step 5 (Input-Edit Profile):

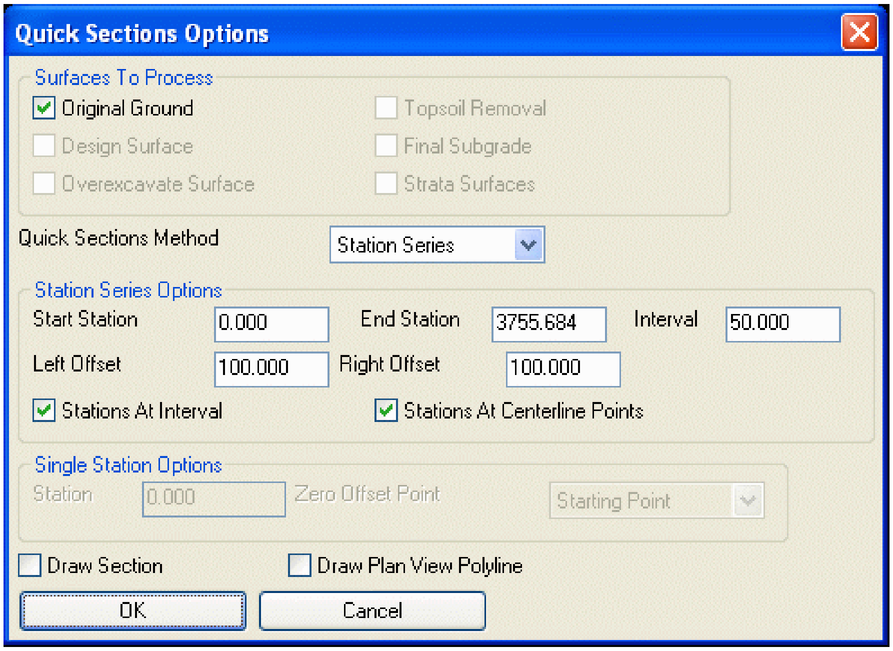
In this routine we will create a Profile file. There are different ways to define a road profile. In this case we will enter values from a given design. Go to Roads-> Input-Edit Profile File, create a new file, and name it Roaddemo. The Input-Edit Profile window will be displayed. Under Type of Profile select Road from the dialog box. In the spreadsheet, you can add design features for the Profile of the road. In this example, enter in the stations 0.0, 1500.0, and 3755.6840, with elevations of 2030, 2005, and 2040. Next, set the Vertical Curve for the middle station at 300.0.



The Slope Percentages and the Sight Distances are automatically computed for you. Select Save and Exit.

Step 6 (Quick Section):

Now we will create the cross section file (∗.SCT). The cross sections define the existing ground for the road to tie into. Run Roads->Quick Section, fill the dialog as shown, and click OK. Your Offsets should be far enough away to tie in the Cut/Fill slopes.



Next, you will be prompted to load a centerline file or to to pick a polyline from the screen. Type C for Centerline and select the demo2.cl file we created earlier. Your section file is now created, your Command line should read as follows:

Command: quicksct

Pick starting point (CL-Centerline,P-Polyline): c

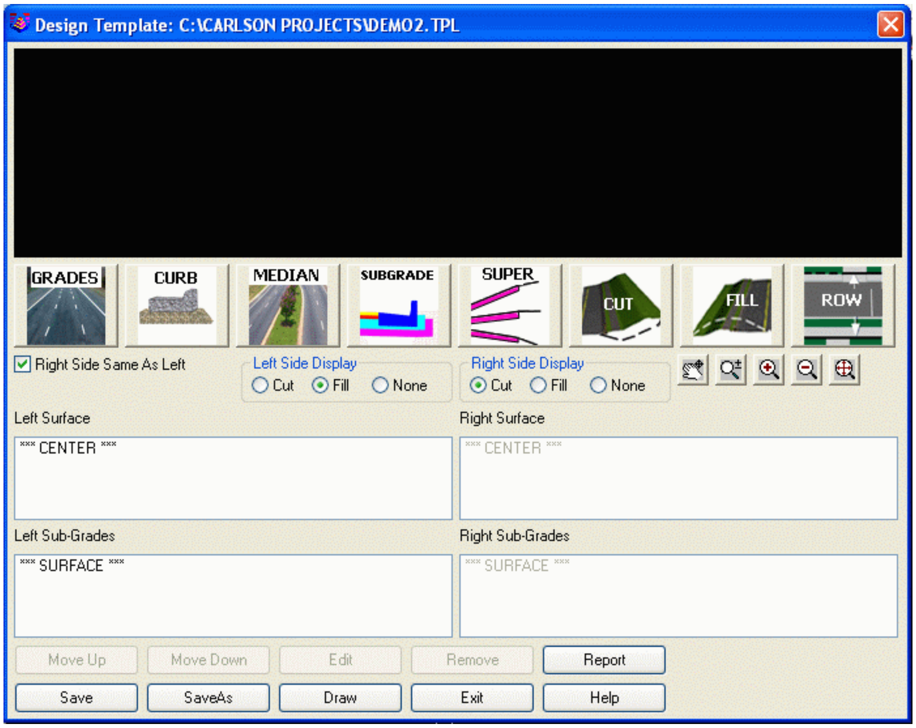
Polyline should have been drawn in direction of increasing stations.
Loading edges...
Loaded 4244 points and 12207 edges
Created 7964 triangles

Writing section C:\Takeoff_2004\demo2a-og.sct

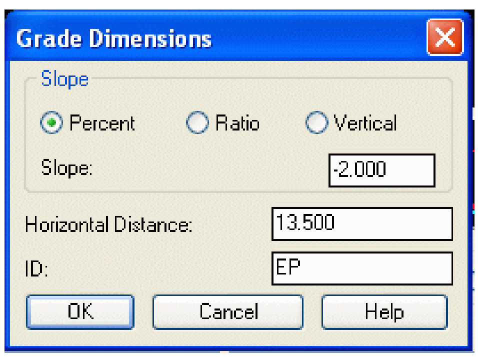
Step 7 (Design Template):

Let's design a wide boulevard with curb and gutter on the outside. The cut slope will be 2:1. In fill, the condition will be 3:1 in all fill under 6' and 2:1 in all fill over 6' in depth. Pavement depths will be 4" of asphalt.

First, select Design Template under Roads, name the .TPL file demo2, and open it. A dialog appears where you enter segments of the template. We will enter a symmetrical template, with 13.5' pavement sections on either side of the centerline, connecting to a 2' curb and gutter, with 18" of gutter and 6" of curb. Then we'll add a 6' shoulder.

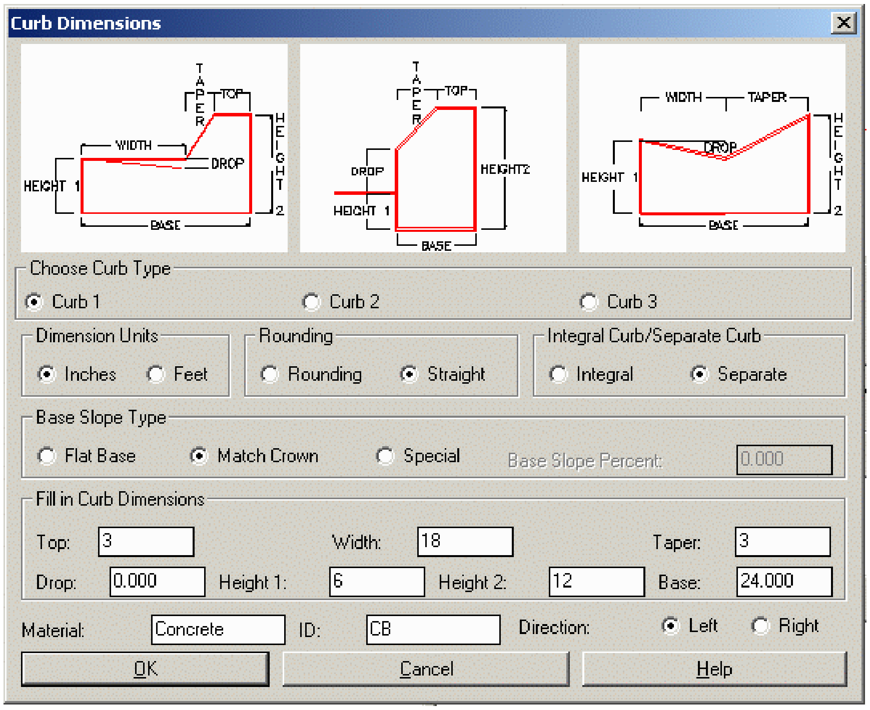


For the lanes, click the Grades Icon.



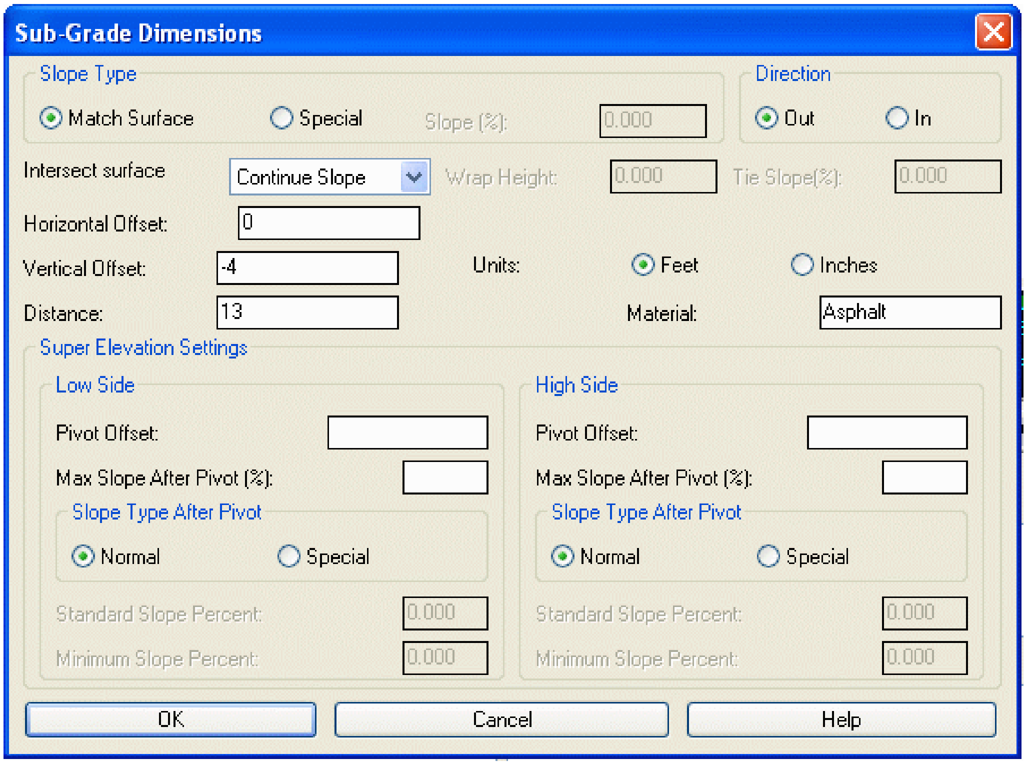
The above 'child' dialog is shown, enter in: Slope: -2, Horizontal Distance: 13.5, and ID: EP. Click OK. You'll note that the lanes draw in the little preview window.

Click on the Curb Icon. Fill out as shown below and click OK.

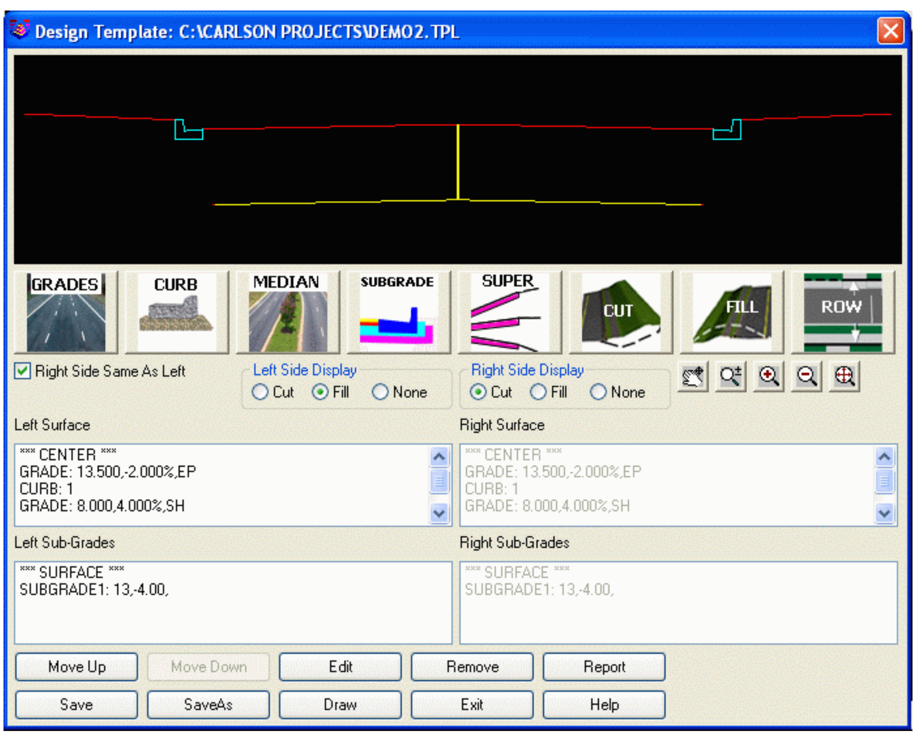


Next up, we will add a shoulder, going up hill at 4% for 8'. Click on Grades again and enter in a slope of 4, a Horizontal Distance of 8, and the ID as SH.

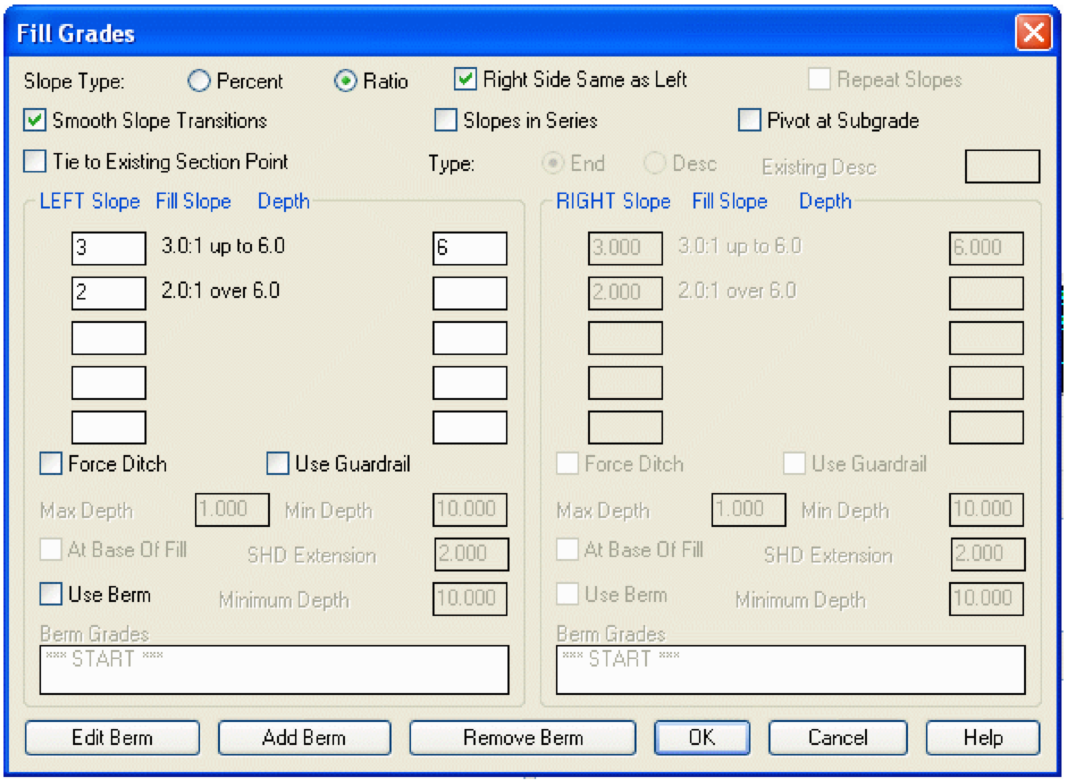
We are now finished with the surface and can set subgrade. Select the Subgrade icon, second from the right (yellow color). We will create an asphalt subgrade which will run straight out and hit the curb.

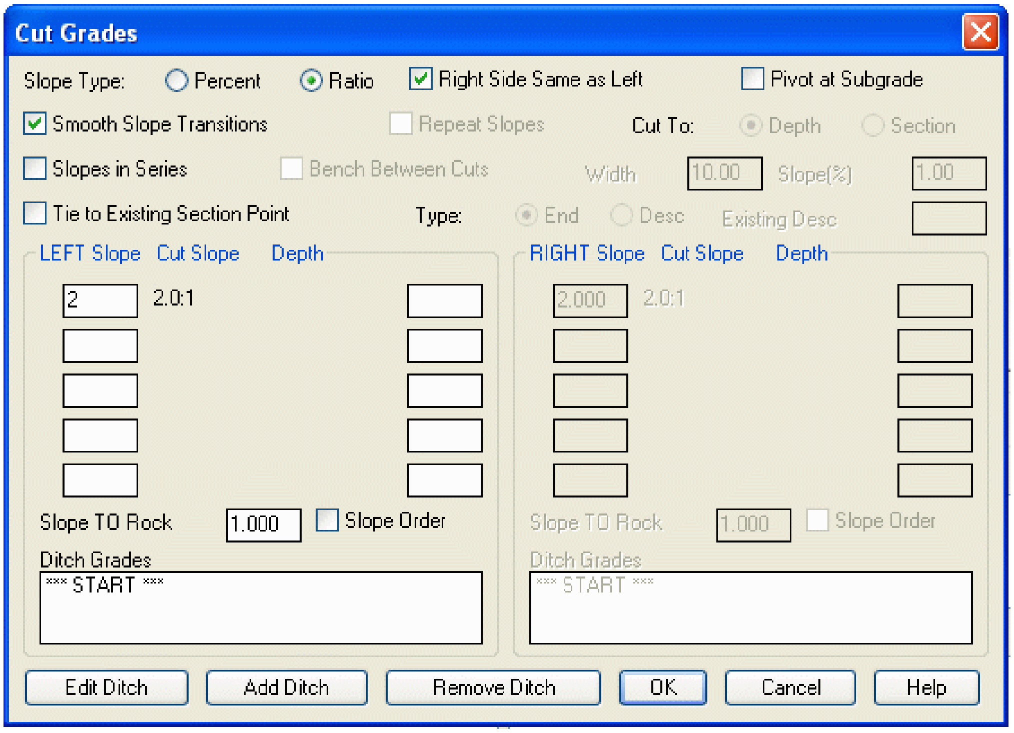


Complete as shown and click OK. Here's what our template looks like so far:



Now we will add the outslope conditions. They are done with the Cut and Fill icons. Click on Fill and make three entries: under LEFT Slope enter in 3 (for 3:1), under Depth enter 6 (up to 6'), then again under LEFT Slope enter in 2 (for 2:1 over 6'). Select OK and click the icon for Cut. Just one entry here: under LEFT Slope enter in 2 (for 2:1 normal cut). Click OK.

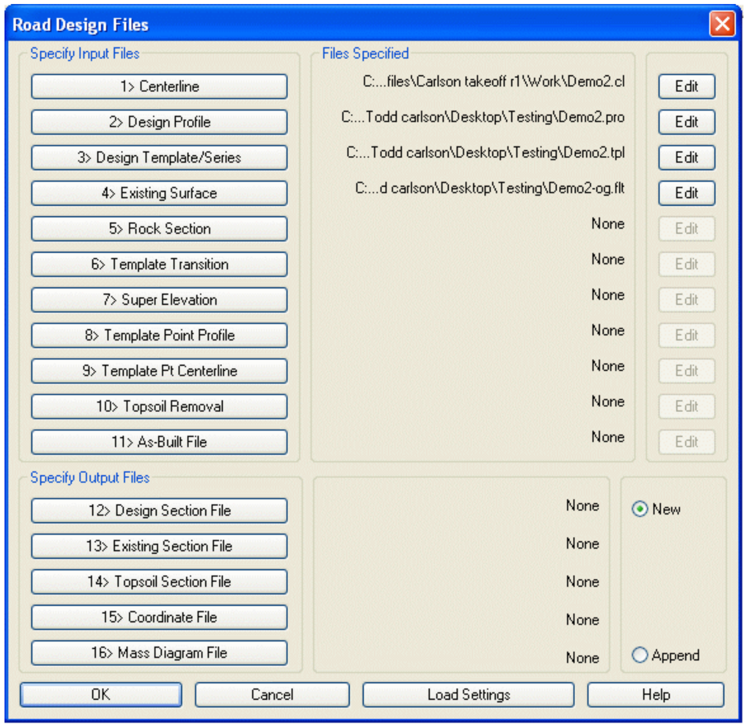




Now click Save. The template is complete.

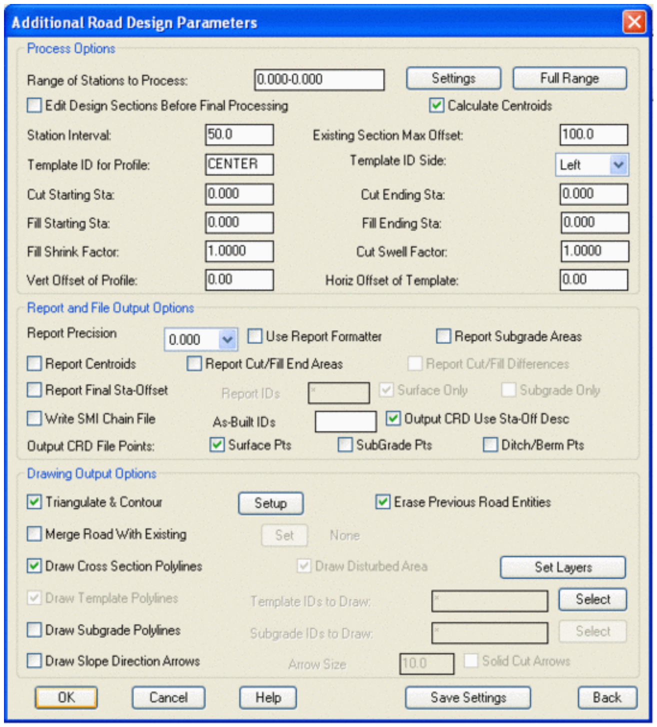
Step 8 (Process Road Design):

This is the routine that weaves everything together. Select Process Road Design, the last command in the Roads menu. The Specify Input Files column on the left allows you to choose the files to be used in the road design. We have already created the four needed files, go ahead and select them now and then click OK.



In the next dialog you can select different output features. For 3D viewing in the next step, toggle on Triangulate & Contour . It can be found in the bottom left portion of the Additional Earth Works Parameters dialog. Click OK.

Note: To generate a plot of road sections, specify an output section file in the 1st Road Design dialog. Then run Draw Sections in the Roads menu.



The following report listing the total Cut, Fill, Subgrade, and Curb volumes.

```
Process Road Design                                          07/29/2004 13:51

Template File> C:\TAKEOFF_2004\DEMO2.TPL
Profile File> C:\TAKEOFF_2004\ROADDEMO.PRO
Existing Surface File> C:\takeoff_2004\demo2a-og.sct
Centerline File> C:\takeoff_2004\demo2.cl

Processing 0+00.000 to 37+55.680
Total Cut : 134177.962 C.F., 4969.554 C.Y.
Total Fill: 2103572.986 C.F., 77910.111 C.Y.
Total SUBGRADE1 - Asphalt: 25357.092 C.F., 939.152 C.Y.
Total CURB - Concrete: 11291.988 C.F., 418.222 C.Y.
```

Trim existing contours inside disturbed area [Yes/<No>]? Press enter to say no

In the Contour Options dialog change the layer name to FINAL ROAD and make the contour interval 2. Click OK and your Road is complete! Here are the Road Design prompts:

Command: eworks

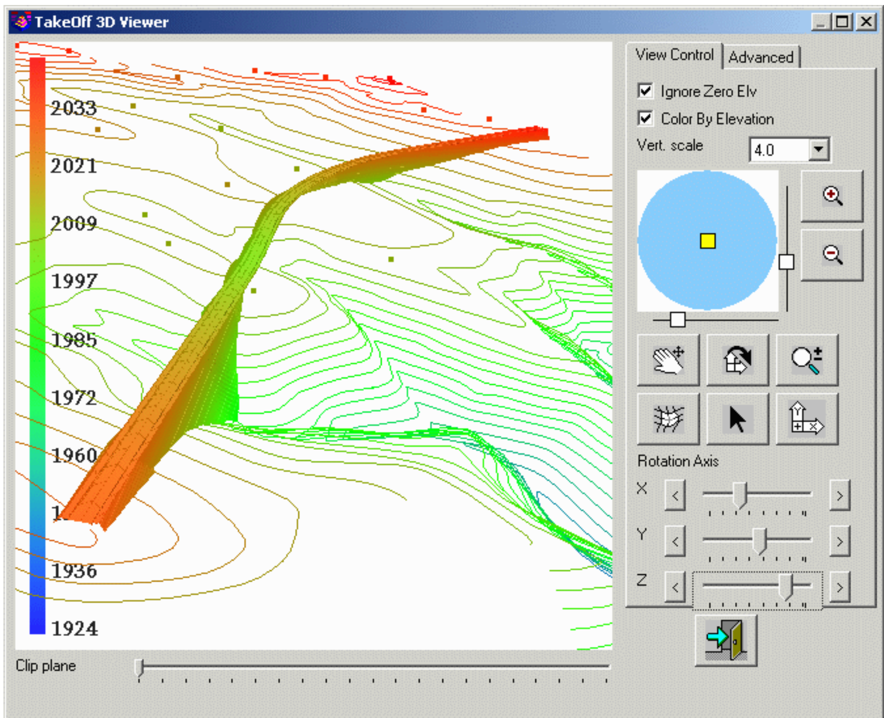
Initializing EarthWorks ...
Processing station: 3755.680
Drawing offset 3D polylines: TIE
Calculating volumes ...
Trim existing contours inside disturbed area [Yes/<No>]? <Enter>

Reading points... 2631
Inserted 2631 points
Inserted 2618 breakline segments

Drawing Triangulation 3D Faces ...
Contouring elevation 2040
Inserted 811 contour vertices.

Step 9 (3D Viewer):

Now that your Road is complete lets view it in 3D. Go to View, 3D Viewer Window, type in ALL, and press enter.



Here is our road with a Vertical Scale of 4. Color By Elevation has also been toggled on. Use the X, Y, and Z control bars at the bottom to rotate the drawing in the 3D viewer.

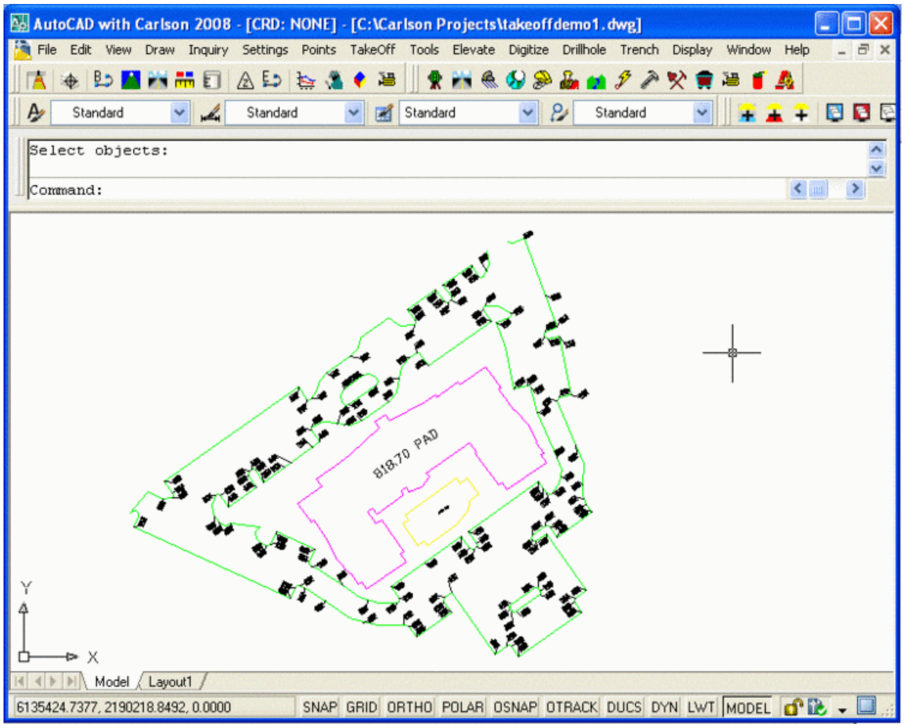
Takeoff Tutorial: Drillholes and Strata

This lesson creates and processes drillhole data.

Step 1 (Run Lesson 1 Example):

This drillhole lesson builds on the resulting drawing called takeoffdemo1.dwg from the tutorial lesson 10 (Takeoff Basics). Before continuing with this tutorial, run through and complete this lesson 1 tutorial.

When lesson 10 is done, let's set the display to show only the design entities. In the Display menu, turn off Existing Drawing and Other Drawing and turn on Design Drawing. Then run, View->Zoom->Extents. Now we're ready to add drillholes.



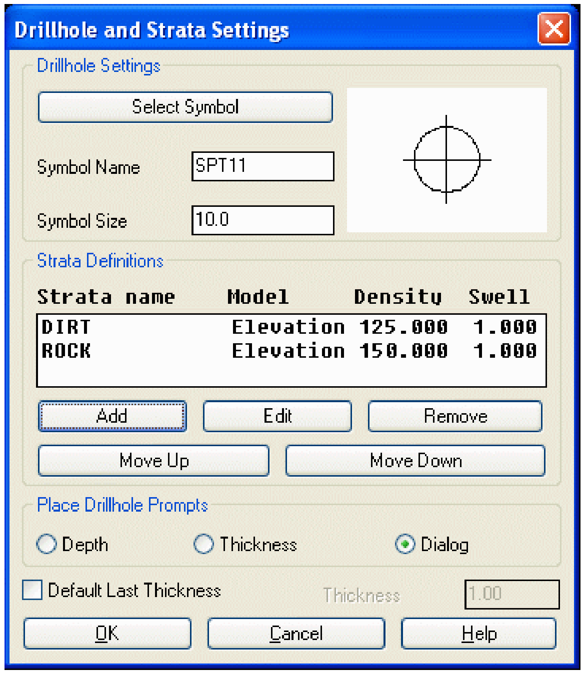
Step 2 (Drillhole/Strata Settings):

From the File menu, choose Drillhole/Strata Settings. This command sets the drillhole symbol and the default strata names. For this tutorial, we are interested in rock quantities and we need to define two strata: Dirt (material above the rock) and Rock.

Pick the Add button which brings up another dialog that defines a strata. Enter a strata name of "DIRT" and a density of 125 which will be used to calculate tons in the volume report. You can also have a strata specific cut swell factor. The strata can be modeled either by the elevations from the drillholes or by the depth from the existing ground. In this case, we will model by strata elevation. When the dialog is filled out as shown, pick OK.

Next, pick the Add button again. This time, fill out the dialog with a strata name of "ROCK" and density of 150. Then pick OK.

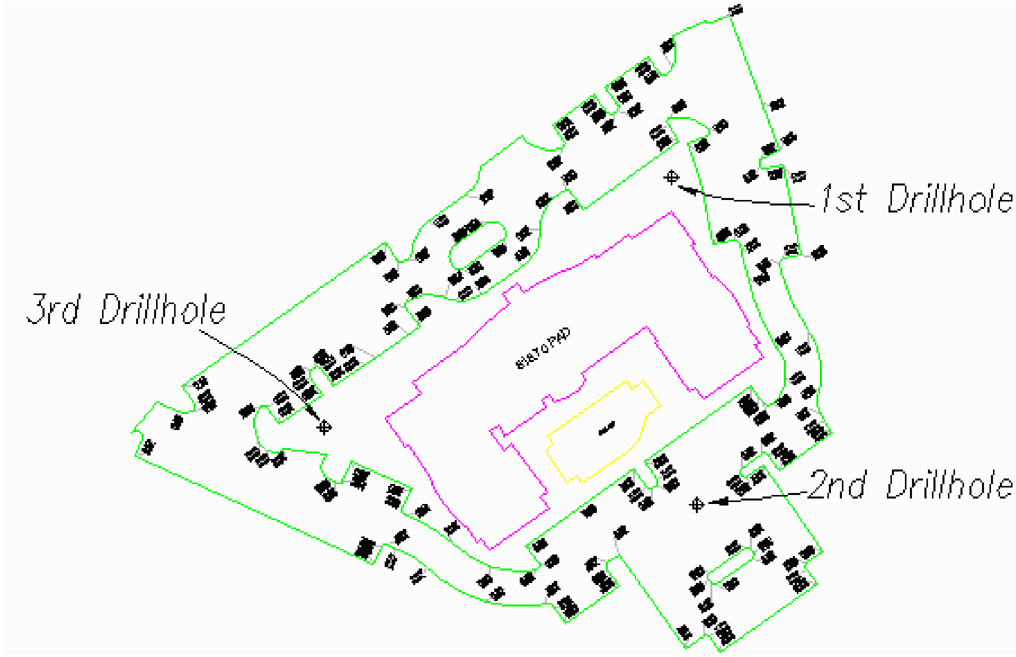
The Strata Definitions in the main dialog need to be in top to bottom order. To change the order, highlight a strata name and use the Move Up or Move Down buttons. In this case, we want Dirt then Rock. Click OK now from the main dialog.



Step 3 (Input Drillhole Data):

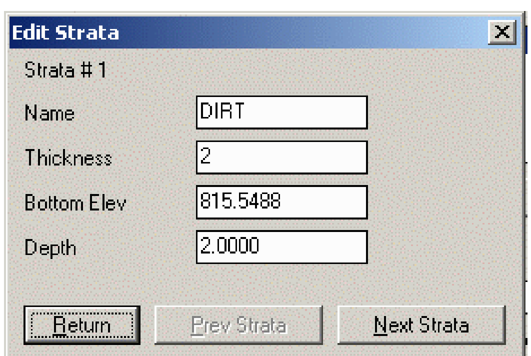
There are two different methods for entering drillhole data into Takeoff: Drillhole Import and Place Drillhole. Drillhole Import reads the drillhole data from a text file. This command supports customizing the sequence of drillhole data fields to match the format of the text file. Place Drillhole creates the drillholes at picked positions in the drawing and enters the data in a dialog. For this tutorial, we will use Place Drillhole.

Run the Drillhole->Place Drillhole command. At the command line, there is a prompt to pick the drillhole location. If you know the coordinates for the drillhole, you can type in the easting,northing instead of picking on the screen. In this case, let's pick a point above the upper right of the main building.
Pick Drillhole Location: *pick a point*



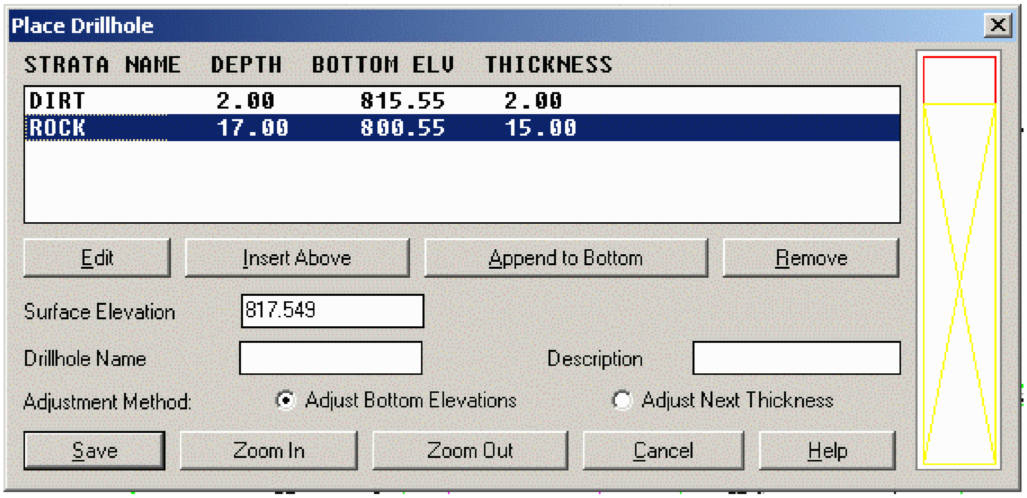
Then there is a dialog for entering the drillhole data. The surface elevation is automatically filled in using the existing ground surface model. The Drillhole Name and Description are optional. The list of strata defaults to the strata defined in Drillhole/Strata Settings. Each strata defaults to a thickness of zero. To set the strata thickness, highlight the strata and pick the Edit button.

For this case, highlight Dirt and pick Edit. This brings up the Edit Strata dialog. The strata position can be defined by thickness, elevation or depth. Setting any one of these fields will update the other fields. For our dirt strata, fill in a thickness of 2 and then pick OK.

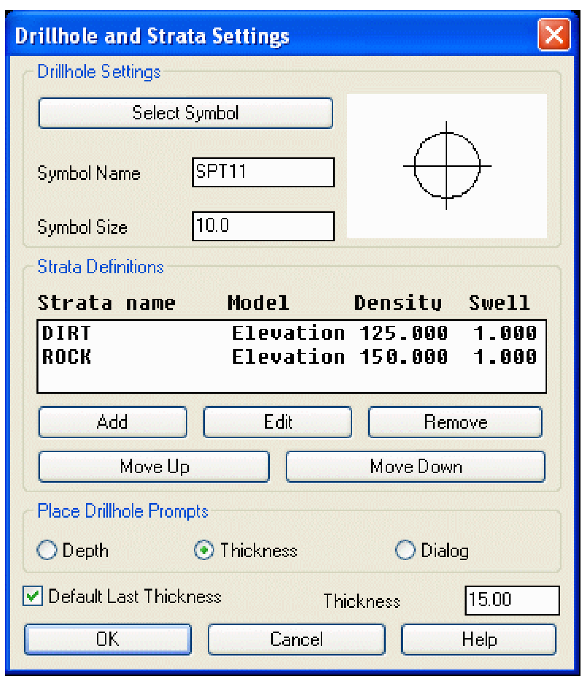


Next, pick Rock from the strata list and pick Edit. For this example, we only know the depth to the top of rock depth and not the total rock thickness. We will treat all cut below the top of rock as rock strata. So we will set the rock thickness deep enough to be lower than the deepest cut on site. In this case, we will use a rock thickness of 15. So in the Edit Strata dialog for rock, enter a thickness of 15 and then pick OK.

After editing the rock strata, we are returned to the main Edit Drillhole dialog. Pick the Save button.



Now let's locate two more drillholes using a different method. Return Drillhole/Strata Settings dialog and change Place Drillhole Prompts to Thickness. Also, check on Default Thickness and set it to 15 feet, Press OK.



Now run Place Drillhole again and for the second drillhole, pick a position in the lower parking lot. The command line will prompt you to enter a dirt thickness, type in 1.5 and your drillhole is created. For the third drillhole, pick a position left of the main building. Enter a dirt thickness of 3.0, save, and enter to end the command.

Step 4 (Make Strata Surfaces):

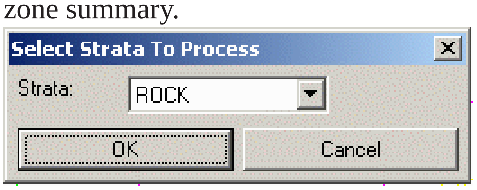
Now that the drillholes are in the drawing, to make the strata triangulation surfaces, run the Drillhole->Make Strata Surfaces command. There are no prompts for this routine. The strata surfaces are modeled from the drillholes and saved with the project. The file names for the strata surfaces use the drawing name plus "-ch#" where the # is the strata sequence number. For this example, the file names will be "takeoffdemo1-ch1" for bottom of dirt and "takeoffdemo1-ch2" for bottom of rock.

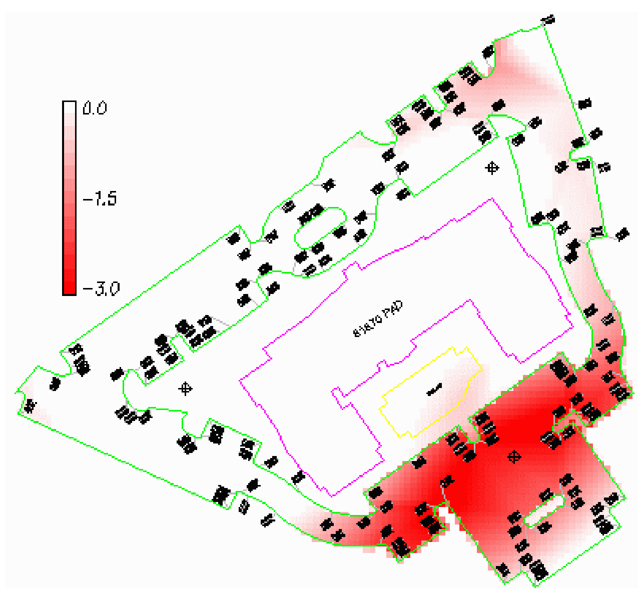
Now that the strata surfaces are created, there are several Takeoff routines that will use these surfaces such as:
- Calculate Total Volumes
- Calculate Volumes Inside Perimeter
- Cut/Fill Labels
- Surface Inspector
- Quick Profile
- Trench Network Quantities

Step 5 (Draw Strata Cut Color Map):

From the Drillhole menu, pick Draw Strata Cut Color Map. This command compares the design surface with the strata surface to make a cut color map of the cut depths for the strata. This command is one way to verify that the strata surfaces are modeled correctly.

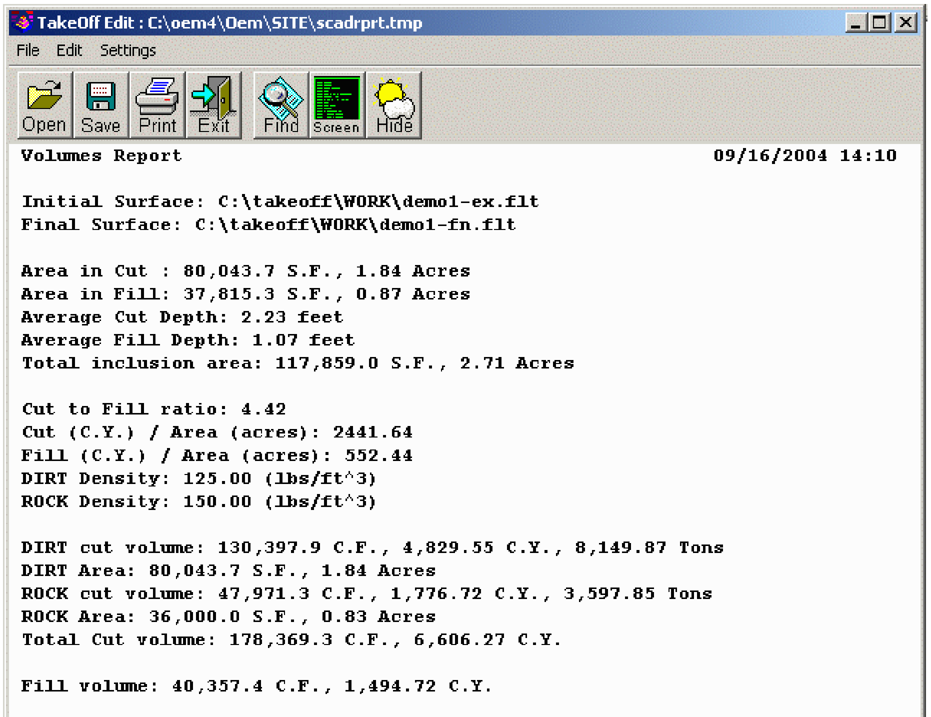
There is a dialog to select which strata map to draw. Choose Rock and pick OK. Then there is an option to draw a cut depth legend. Pick a position for the legend in the upper left of the site and use the defaults for size and zone summary.





Step 6 (Calculate Total Volumes):

Run the Takeoff->Calculate Total Volumes command. When strata surfaces are defined, the volume routine will breakout the cut volume into the different strata. The resulting dirt and rock quantities are shown in the report.



Takeoff Tutorial: Trench Network Quantities

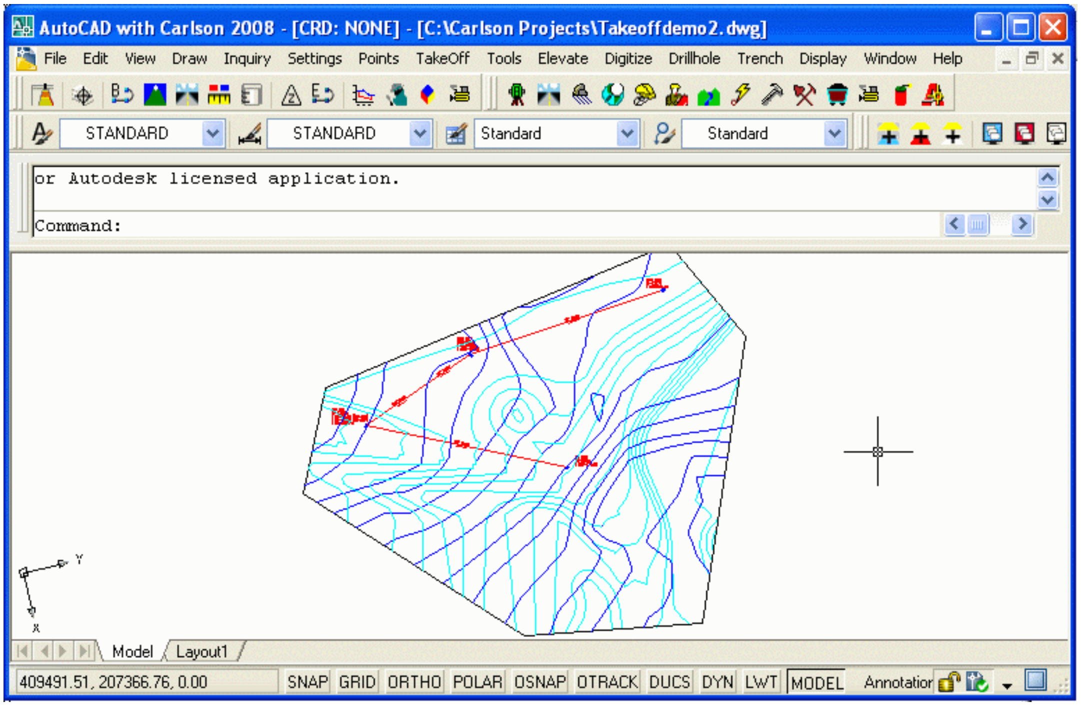
This lesson takes a drawing file through the steps of trench network quantities.

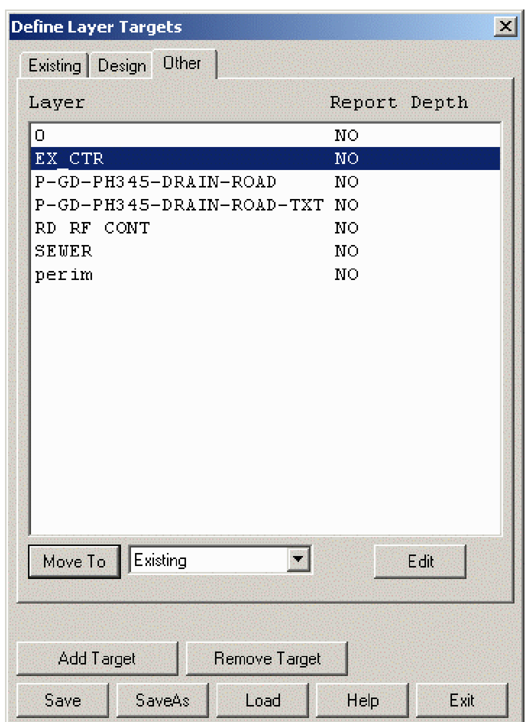
Step 1 (Start Takeoff):

Click the icon for Takeoff on your desktop or from the toolbar to launch the program. You may be presented with a "Startup Wizard" dialog and if so, click Exit.

Step 2 (Open Drawing):

From the File menu, choose Open and select Takeoffdemo2.dwg from the Carlson Projects folder.





Step 3 (Make Existing and Design Surfaces):

In order to calculate trench quantities and profiles in this drawing, we need surfaces for the existing ground and design.

First we need to define the layers of the surfaces. Run TakeOff->Define Layer Target/Material/Subgrade. Then from the tab labeled "Other", highlight "EX_CTR" from the layer list, pick "Existing" from the Move To list and pick the Move To button. Next, highlight "RD RF CONT", choose "Design" from the Move To list and pick the Move To button. Now choose the Save and then Exit buttons. This assigned layer "EX_CTR" to the existing ground surface and "RD RF CONT" to the design surface.

Next, let's set the site perimeter. Run TakeOff->Boundary Polyline->Set Boundary Polyline. At the command line, there is a prompt:
Select boundary polyline:
Pick anywhere along the six sided perimeter polyline in the drawing.

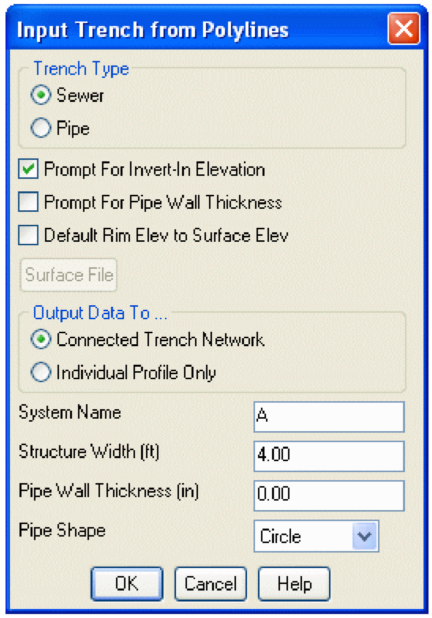
Now, to make the existing ground surface, run TakeOff->Make Existing Ground Surface. Then to make the design surface, run TakeOff->Make Design Surface.

Step 4 (Input Trench Network Data):

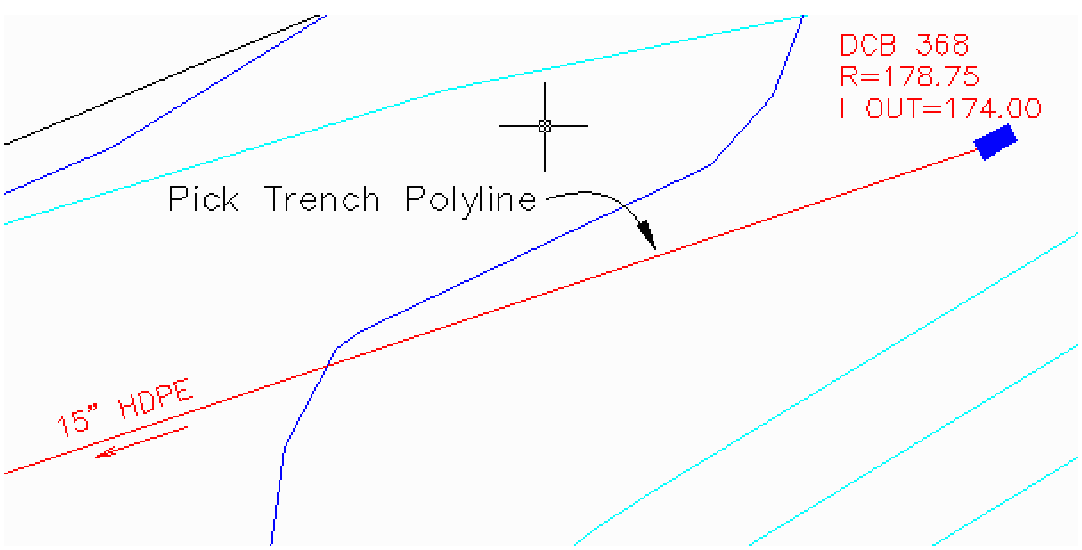
The trench network data consists of linked structures where each structure has a name, location (x,y), invert-in, invert-out and rim elevation. Each structure link has a pipe size. There are two ways of entering the trench data. When the drawing contains polylines for the trench lines and labels with the trench data, then you can use Input Trench From Polyline. Otherwise, there is the Create Trench Network Structure command which let's you pick the structure locations and enter the data in a dialog.

Method 1 (Input Trench Data From Polyline):
In this example, there is trench data already drawn in the drawing. Zoom in around the upper right area of trench line by running View->Zoom->Window and picking two corner points around this area.

Then run Trench->Input Trench From Polyline and an options dialog appears. In this case, we want Trench Type as Sewer because there are manhole rim elevations. Also Prompt For Invert-In Elevations is active since this example has a manhole with multiple connections with different invert-ins. And Connected Network is used so that the trench data can be used by the rest of the trench routines. The Individual Profile option will only create a profile (.pro) file. Prompt For Pipe Wall Thickness allows you to enter in the pipe thickness that will be added to the interior pipe size for accurate volume calculations. Fill out the dialog as shown and click OK.



The rest of the prompting for this command is on the command line as the program walks through the trench line. For each point in the trench polyline, the program zooms the drawing to that point. The trench data can be picked from labels in the drawing. If the drawing doesn't have labels for the data, then you can enter the values.

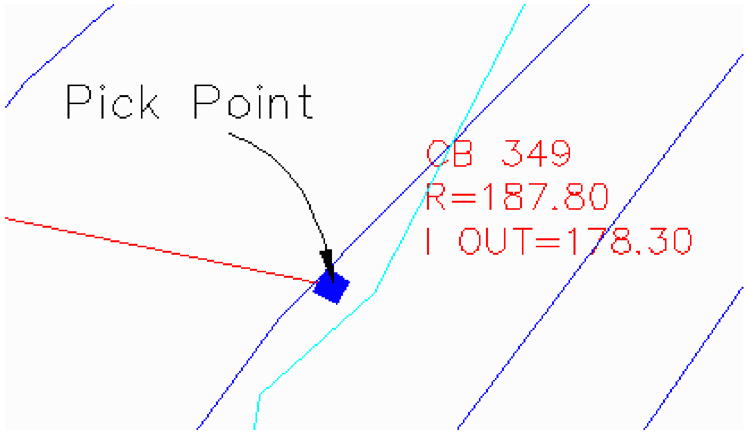


Pick a polyline that represents a trench reach: *Pick the trench polyline*
Starting Station of trench reach <0.0>: *0.0*
For station 0.00 ...
Enter/<Select text of Manhole ID>: *Pick the DCB 368 label.* (If you had a drawing without a manhole ID label, then type E for Enter and enter the ID)
ID: DCB 368
Undo/Enter/<Select text of Invert-in elevation>: *Pick the I Out=174 label.* (Since this is the upstream starting

manhole, there really isn't a separate invert-in. So we are using the invert-out).
Invert-In: 174.000
Undo/Enter/<Select text of Invert-out elevation>: *Pick the I Out=174 label.*
Invert-Out: 174.000
Undo/Enter/<Select text of manhole rim elevation>: *Pick the R=178.75 label.*
Rim: 178.750
For station 201.44 ...
Enter/<Select text of Manhole ID>: *Pick the DCB 367 label.*
ID: DCB 367
Undo/Enter/<Select text of Invert-in elevation>: *Pick the I In=172.85 label.*
Invert-In: 172.850
Undo/Enter/<Select text of Invert-out elevation>: *Pick the I Out=172.35 label.*
Invert-Out: 172.350
Undo/Enter/<Select text of manhole rim elevation>: *Pick the R=178.5 label.*
Rim: 178.500
Undo/Enter/<Select text of pipe size>: *Pick the 15'' HDPE label.*
Pipe Size: 15.0
For station 327.09 ...
Enter/<Select text of Manhole ID>: *Pick the CB 347 label.*
ID: CB 347
Undo/Enter/<Select text of Invert-in elevation>: *Pick the I In=170.540 (CB 367) label.* (This is the invert-in for the connection to the CB 367 structure that this trench line connects to.)
Invert-In: 170.540
Undo/Enter/<Select text of Invert-out elevation>: *Pick the I Out=166.1 label.*
Invert-Out: 166.100
Undo/Enter/<Select text of manhole rim elevation>: *Pick the R=176.5 label.*
Rim: 176.500
Undo/Enter/<Select text of pipe size>: *Pick the 15'' HDPE label.*
Pipe Size: 15.0
Another Polyline [<Yes>/No]? *N* for no.

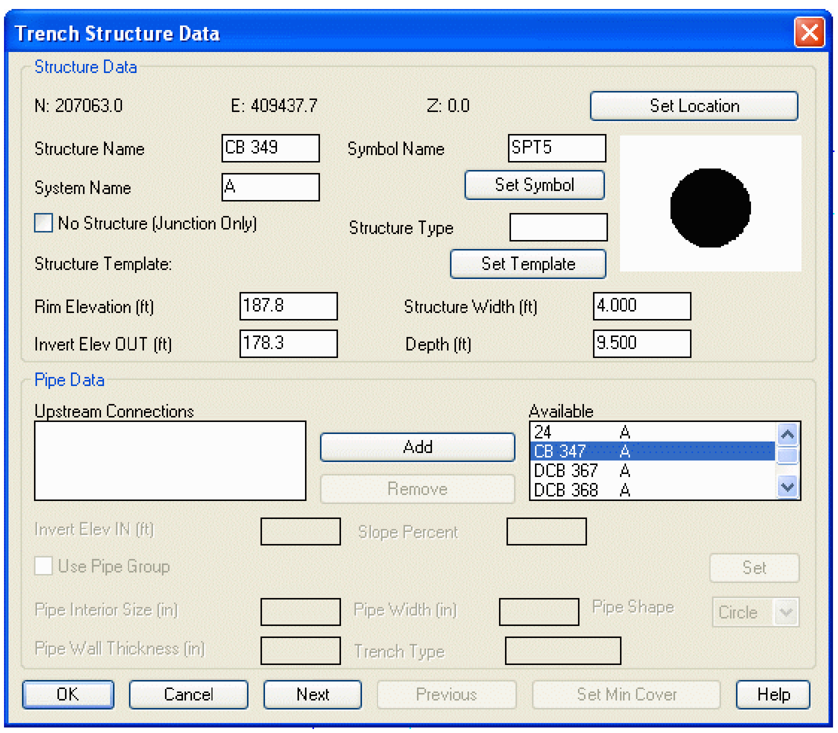
That completes this trench run and Takeoff draws its own trench polyline and labels.

Method 2 (Create Trench Network Structure):
The drawing contains another trench polyline and we could use Input Trench Data From Polyline again. Instead for practice, let's use the Create Trench Network Structure method. First we need to zoom to the new trench location. Run View->Zoom->Extents and then View->Zoom->Window and pick two points for a window around the lower right trench point (CB 349). Then run Trench->Create Trench Network Structure. At the command line, there is a prompt for how to locate the structure position. Choose Pick.
Locate by pick point, point number or station-offset [<Pick>/Number/CL]? *Pick*



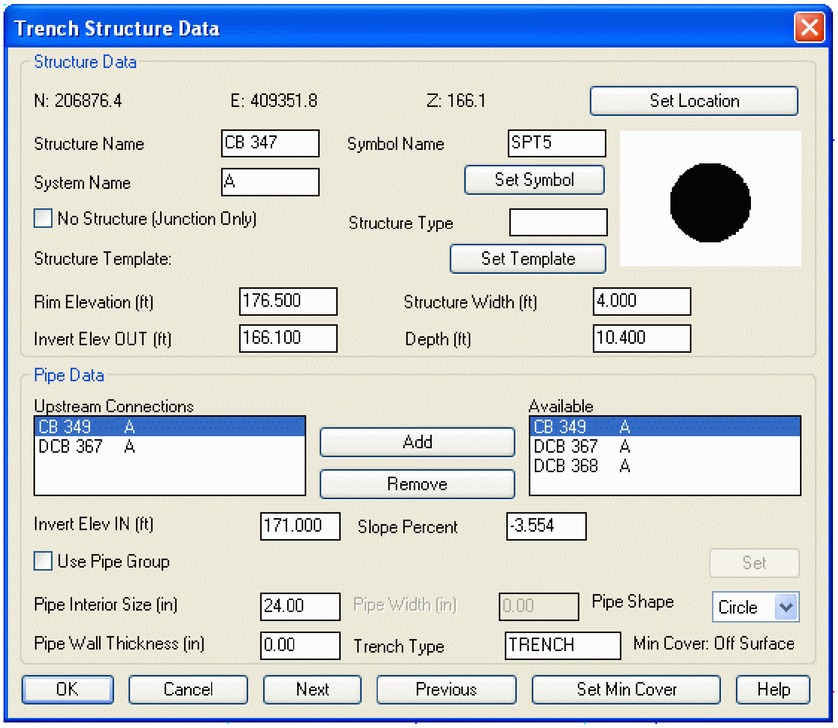
Next, there is a prompt to pick the position. To get the exact end point of the trench polyline, use the end point snap. The end point snap can be turned on by a number of different ways including the Settings->Object Snap command. In this case, type "end" and then space or enter. This puts the program in end point snap mode. Now move the pointer along the trench polyline until the end point snap icon is at the manhole location and then pick.
Pick structure location: *end of (pick point)*

Now a dialog appears for entering the structure data. Fill in the Structure Name as CB 349, the Rim Elevation as 187.8, the Invert-Out as 178.3, the Structure Width as 4.00 and then pick OK.



All the structures are now created. The last step is to link this new structure to the network. We need to zoom to the next trench location. Run View->Zoom->Extents and then View->Zoom->Window and pick two

points for a window around the left trench point (CB 347). Now run Trench->Edit Trench Network Structure and pick either the symbol for CB 347 or the label. Then a dialog appears with the data for CB 347. From the Available list, highlight CB 349 and pick Add. This creates a link from CB 347 to CB 349 and the link data is shown at the bottom of the dialog. Enter the Invert-In as 171 and the Pipe Size as 24. Then pick OK.

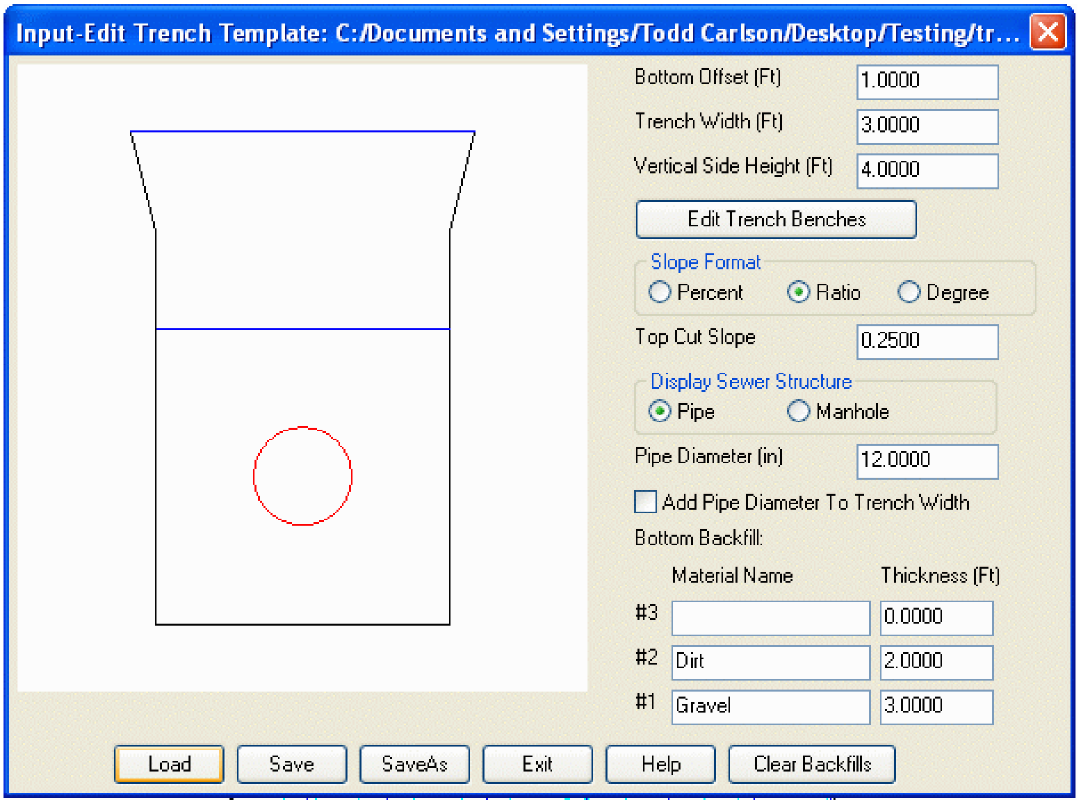


Step 5 (Input-Edit Trench Template):

The Trench Template defines the size of the trench for quantities. Run Trench->Input-Edit Trench Template. You are first prompted for a trench template file name. The Trench Template data is stored in a file that has a .tch extension. Choose the New tab, enter a file name like trench1 and then pick Open.

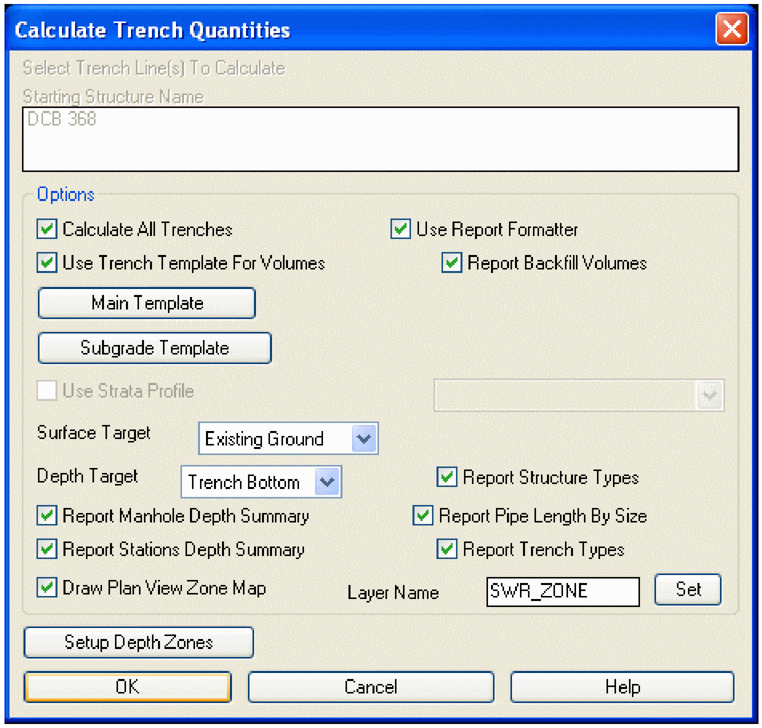
Next, there is a dialog for entering the trench dimensions. The Bottom Offset is the distance from the bottom of the pipe to the bottom of the trench. The Trench Width is the base width of the trench. The Vertical Side Height is the height from the bottom that the side walls are vertical until switching to the cut slope. If the surface is not reached by the vertical side height, then the cut slope is used for the rest of the distance to the surface. Edit Trench Benches allows you to set up to four benches in your trench. Display Sewer Structure allows you to see your pipe or manhole as part of the trench. Note: This is for display purposes only, calculations will be drawn from the pipe size you set in the Trench Network Structure commands. Add Pipe Diameter To Trench Width will increase the size of your trench by the diameter of your different pipe sizes. The Cut Slope can be entered in slope percent, ratio or degree format. The Backfill materials are optional. Up to three materials can be entered from the bottom.

Fill out the dialog as shown and pick Save and Exit.

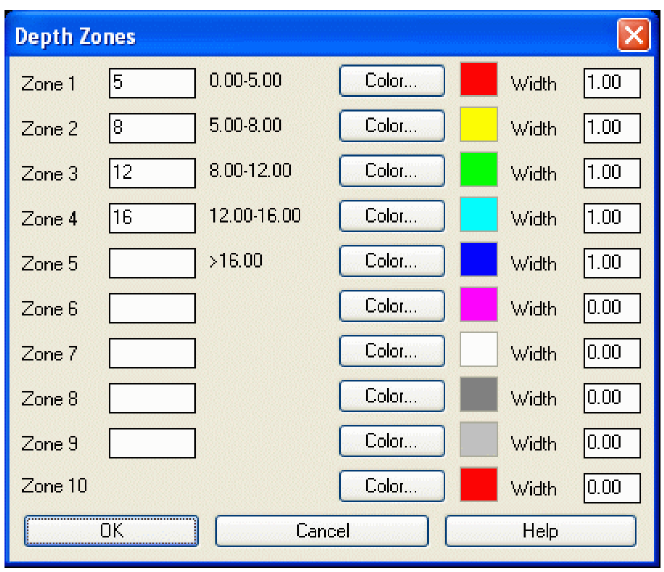


Step 6 (Trench Network Quantities):

To calculate and report the trench quantities, run Trench->Trench Network Quantities. A dialog sets the report options. Check on Calculate All Trenches to get the quantities for the whole network. To get the trench cut volume, check Use Trench Template For Quantities and pick Set Trench Template and pick trench1.tch. Also turn on Report Backfill Volumes to use our backfill material settings from the trench template. Finally, fill out the depth zones in the intervals that you are interested in. In this case, use 15, 20 and 25. The depth zones will be colored in the plain view. Once the dialog is filled out as shown, pick Setup Depth Zones.

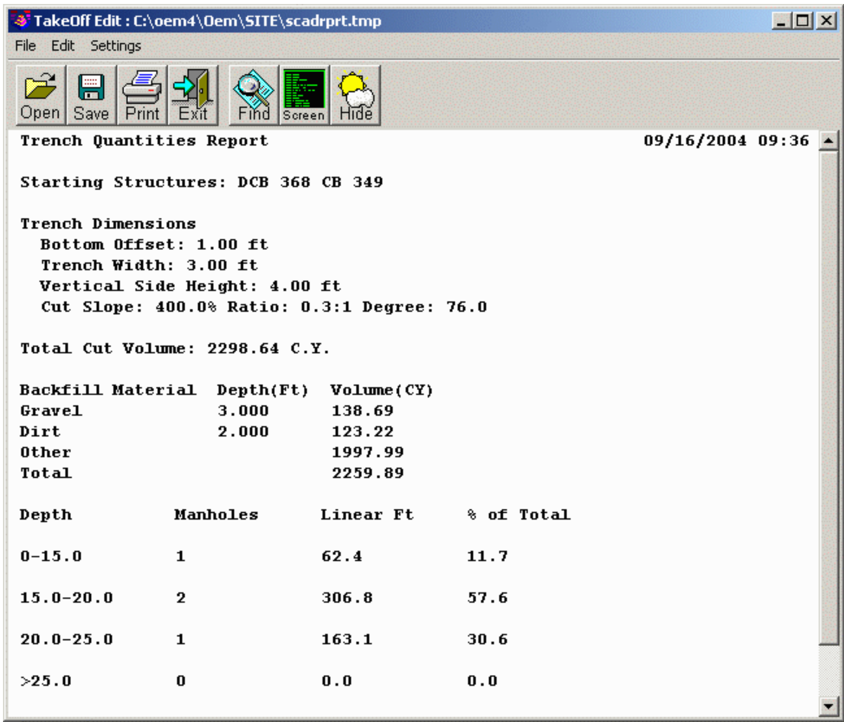


Fill out the depth zones in the intervals that you are interested in. In this case, use 15, 20 and 25. The depth zones will be colored in the plain view. Finally, click OK and OK in the main dialog and the report is shown.



The report includes:
- The structure names at the start of each trench run included in the report.
- The trench template dimensions.

- The cut volume.
- The backfill volumes.
- The number of manholes and length of trench within each depth zone.



Trench Quantities Report 09/16/2004 09:36

Starting Structures: DCB 368 CB 349

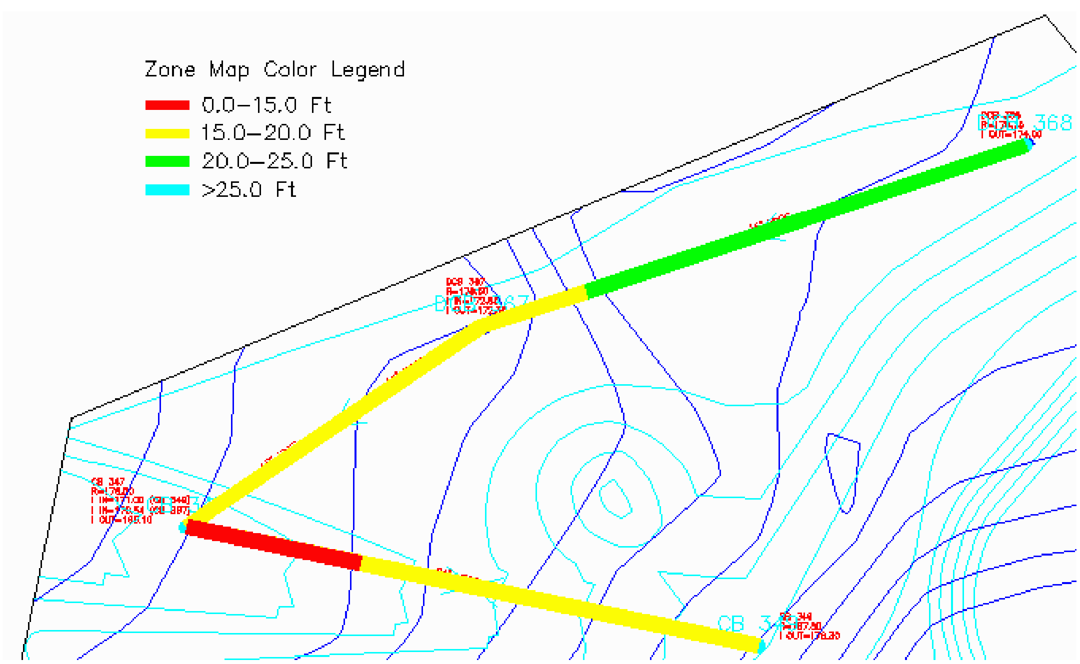
Trench Dimensions
Bottom Offset: 1.00 ft
Trench Width: 3.00 ft
Vertical Side Height: 4.00 ft
Cut Slope: 400.0% Ratio: 0.3:1 Degree: 76.0

Total Cut Volume: 2298.64 C.Y.

Backfill Material	Depth(Ft)	Volume(CY)
Gravel	3.000	138.69
Dirt	2.000	123.22
Other		1997.99
Total		2259.89

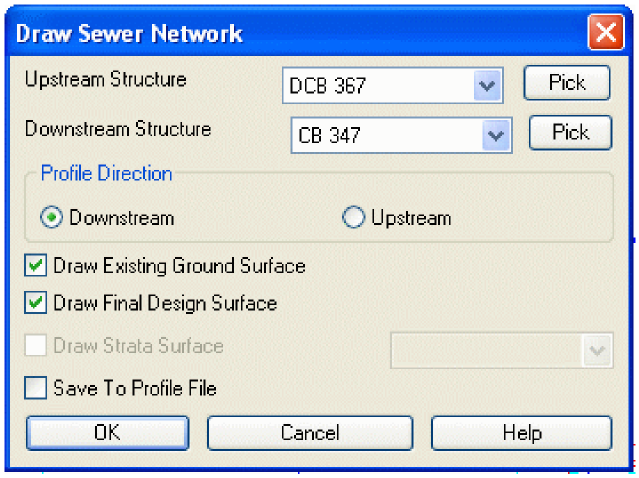
Depth	Manholes	Linear Ft	% of Total
0-15.0	1	62.4	11.7
15.0-20.0	2	306.8	57.6
20.0-25.0	1	163.1	30.6
>25.0	0	0.0	0.0

The depth zones in the plain view with Zone Map Color Legend.

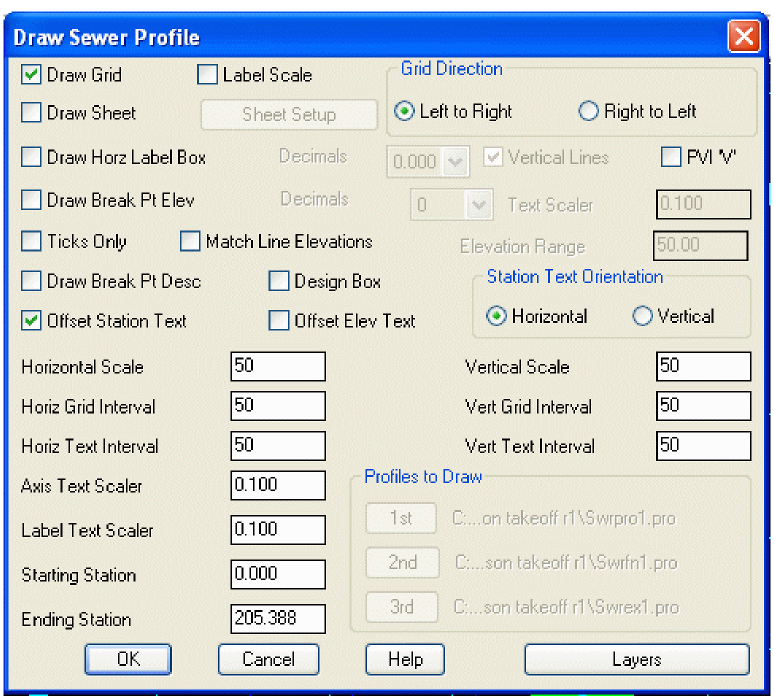


Step 8 (Draw Trench Network Profile):

To draw a profile of the trench line, run the Trench->Draw Trench Network (Profile) command. There is a dialog to select the starting structure for which trench line to process. Choose DCB 368. Also, there are options whether to draw the existing ground, design surface and strata surfaces if available. You can also choose the profile direction to go upstream or downstream. The Save To Profile File will create a profile (.pro) file for the trench. Fill out the dialog as shown and pick OK.

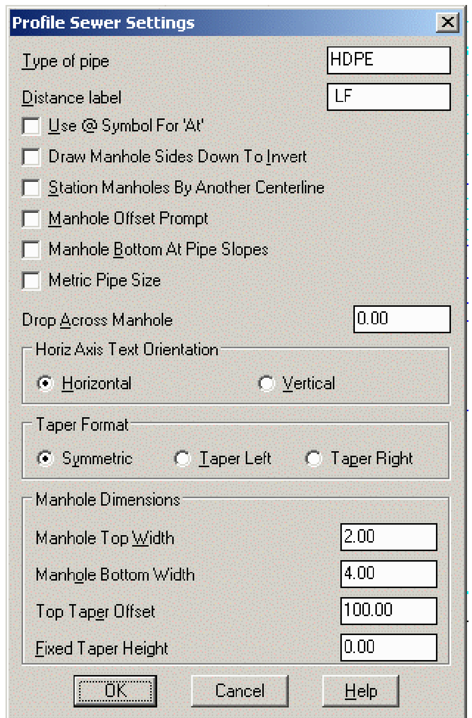


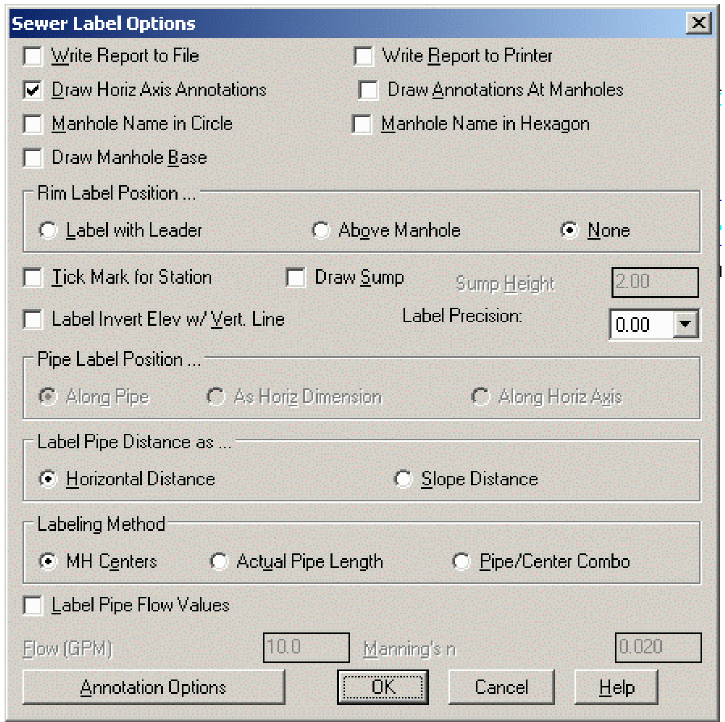
Next the Draw Profile dialog appears. Set the Horizontal Scale and intervals to 50 and the Vertical Scale and intervals to 25. This will make for two to one vertical exaggeration for the profile. When the dialog is filled out as shown, pick OK.



Next, there are prompts at the command line for the profile grid elevations and profile location.
Bottom Elevation of Profile Grid <150.0>: *Press Enter*
Top Elevation of Profile Grid <200.0>: *Press Enter*
Pick Starting Point for Grid <409458.0 , 207303.0>: *Pick a point in a blank area off to the side of the drawing.*

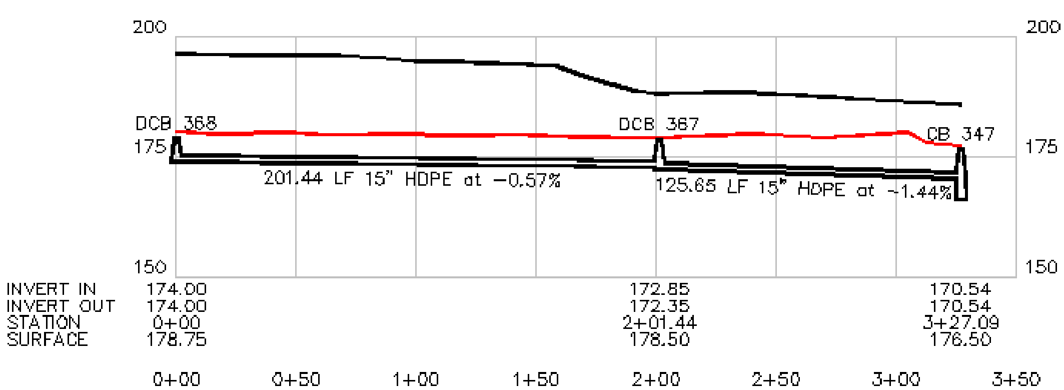
Next there are two dialogs with lots of settings for how to draw and label the trench profile. Let's go with the defaults except change the Type Of Pipe to HDPE. When the settings are ready, click the OK button for each dialog.





There is a final command line prompt for whether to use the manhole elevations. Enter Yes which will use the rim elevations defined in our trench network.
Use manhole elevations from profile [<Yes>/No]? *Y* for yes.

The profile shows the existing ground at the top, then the design surface and then the trench.



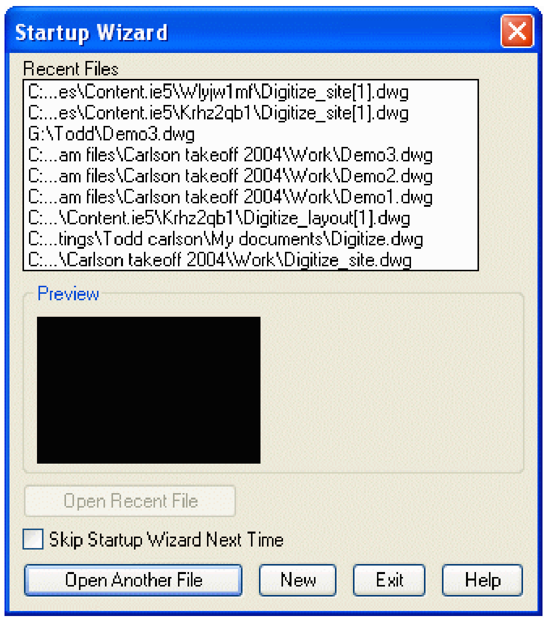
Takeoff Tutorial: Digitizing

This lesson transfers a paper plan into Carlson Civil.

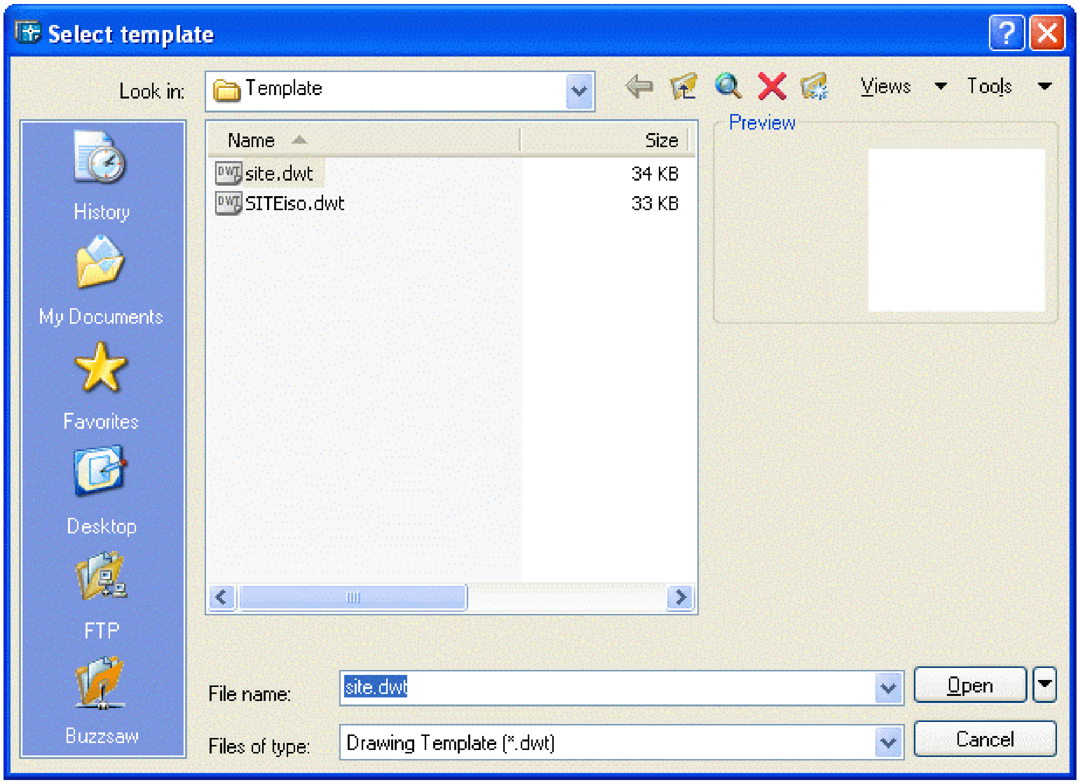
Step 1 (Setup):

To digitize in Carlson Takeoff, you need to install the Wintab digitizer driver. See Digitizer Setup in the manual if you have not installed or have problems with the Wintab driver. If Wintab is installed, then make sure your drawing board is on and take the paper plan provided with the manual and place it on your drawing board.

Click the icon for Takeoff on your desktop or from the toolbar to launch the program. You may be presented with a "Startup Wizard" dialog similar to the one shown below, if so click New.



If a Startup Wizard did not appear, then under File menu, select New to start a new drawing. You will be prompted for a template to use. Templates determine the default settings for your drawing. For this tutorial, select site.dwt and click Open.

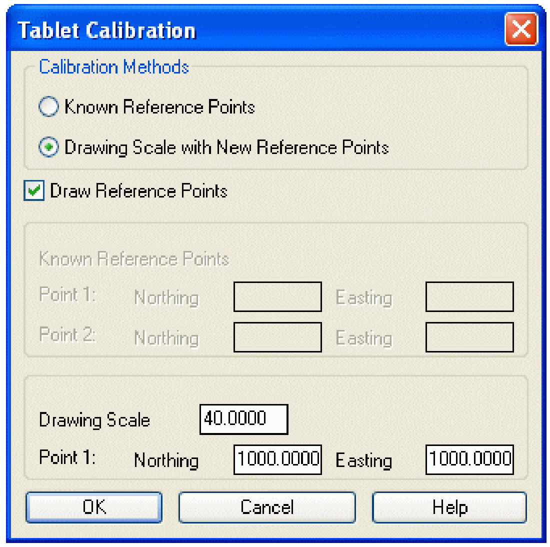


Next, the New Drawing Wizard appears for setting the drawing name. Click on the Set button at the top dialog. In the file selection dialog, enter the file name of "digitize" and pick the Save button. Then Exit the New Drawing Wizard. From here, a Data Files dialog appears where no changes are needed. Pick the Exit button.

Step 2 (Tablet Calibration):

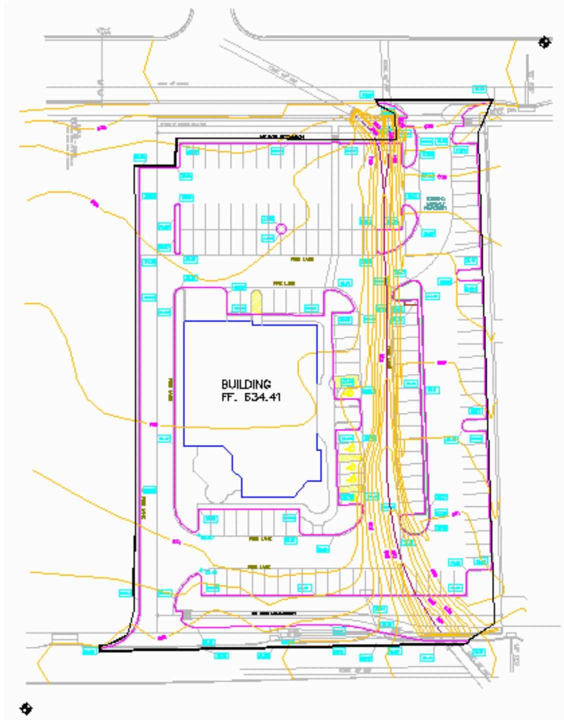
To start things off, you need to set the coordinate system for the paper plan by running the Calibrate command under Digitize menu and sub-menu Tablet. Calibration is required to let the program know the orientation and scale

of the paper plan.



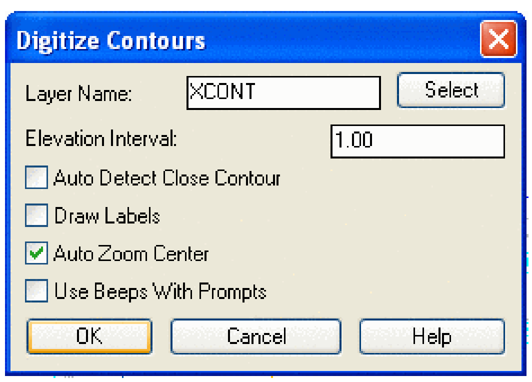
There are two different Calibration Methods: Known Reference Points and Drawing Scale with New Reference Points. Known Reference Points allows you to enter in the coordinates of two marked points on the paper plan. This method applies when you know the coordinates of at least two points on the paper plans. Drawing Scale with New Reference Points allows you to setup a coordinate system for the plans by entering the plan scale and picking any two points from the paper plan with the digitizer puck.

In this case, we will use Drawing Scale with New Reference Points. First, enter in the Drawing Scale listed on the paper plan. On this drawing, the scale is 1:40, so enter in 40. Use the default coordinates for Point 1 and click OK. Now Carlson Takeoff will prompt you for your First and Second Reference points. Generally, you want to pick to points on the drawing that you can find and use again in case you need to recalibrate. Also, the further away the points are from each other, the more accurate the coordinate system will be. With the digitizer puck, pick on the  icon in the lower left and upper right of the drawing for the two Reference Points. The first point is assigned the coordinates of 1000,1000 from the dialog and the second point is assigned coordinates to match with the plan scale. From now on, all of your points will be in relation to these two points.

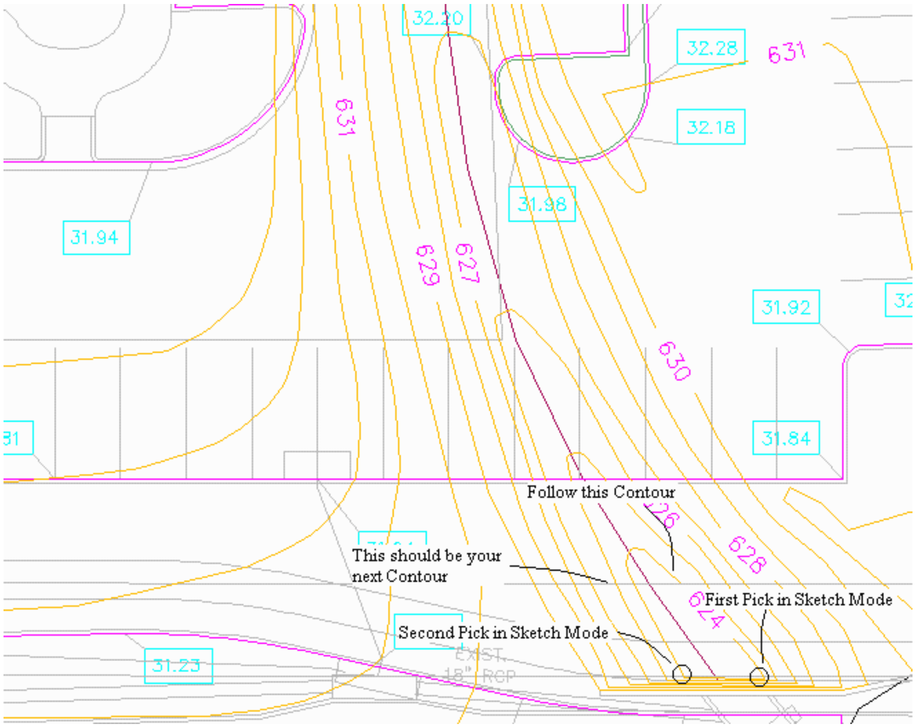


Step 3 (Digitizing Existing Contours):

We will now digitize the existing contours. Under the Digitize menu, click on Existing and then go to Contour Polyline, and this dialog will appear. Enter in a Layer Name of XCONT and select OK. Note: your Elevation Interval should match the intervals marked on your paper drawing. In this drawing, the interval is the same as the default of 1.00.



The rest of the prompting occurs at the command line and starts with the contour elevation. Find the lowest elevation for the existing contours labeled in bottom right corner of the paper plan zoomed in on below. In this example, the lowest elevation is 624 feet. The elevation can be entered either with the digitizer puck keys or with the computer keyboard. The layout of the digitizer keys is set in Digitizing Settings->Puck Layout. Press Enter after you have entered in 624. You want to enter in the lowest contour so that as Carlson Takeoff adds the Elevation Interval, it is from lowest to highest.



Next, you will see the following prompt:

Sketch[0]/Exit[A]/Pick the first point:

There are two different ways to digitize: in Pick Mode or Sketch Mode. You can switch between them at anytime. In this tutorial we will run through how to do both. For now, type in [0] and press enter to get into Sketch Mode. In Sketch Mode, you will be prompted to Pick and drag. The point you pick is the starting point of a contour. Drag is asking you to follow that contour with the digitizer puck on the paper plan. Click a second time when you have traced the entire contour and have reached the end of the contour. You will then be prompted as follows:

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end):

Type in [B] for Undo if you made a mistake and need to sketch part of the contour again. [A] will close the contour, and [0] will switch you into Pick Mode. We still have more existing contours to digitize, so press Enter to end and answer yes to the Digitize Another Contour prompt. Takeoff will prompt you to verify the elevation. Remember, we set the Elevation Interval to one, so the default elevation for your next contour line is 625, press Enter. Now, pick the endpoint of the next contour and trace it in the same manner as the previous contour.

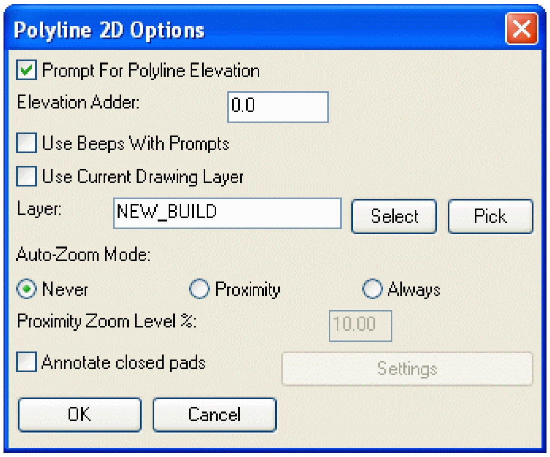
Now let's try Pick Mode. Say yes to digitize another contour and check to see if the default elevation corresponds with the contour your about to digitize. If not, simply type in the correct number in the command line. Next, pick [0] to get into Pick Mode. In Pick Mode, you do not have to trace the contour. Rather, pick with the digitizer puck to create points that will make up the contour. Note: Less picks are needed on fairly straight segments. Conversely, more picks will give you a more accurate contour. Press Enter when you have reach the end of the

contour. Repeat this until you have digitized all of the existing contours you want to have in Takeoff (see below).

Step 4 (Digitizing the Design):

Now we will digitize the building and curb linework of the Design Surface using the Digitize 2D Polyline and 3D Polyline commands. Besides drawing the linework positions, we will also assign layer names to the linework that we will use later to identify the types of linework. In this example, there are no design contours, only the design building and curb linework and spot elevations.

Let's begin by digitizing the main building. Under Digitize, check on Design and go to 2D Polyline. 2D Polyline is used to digitize linework entities with one elevation. Toggle off the check box **Use current drawing layer** and name the layer NEW BUILD. Toggle on the **Prompt For Polyline Elevation** option. Then click OK.

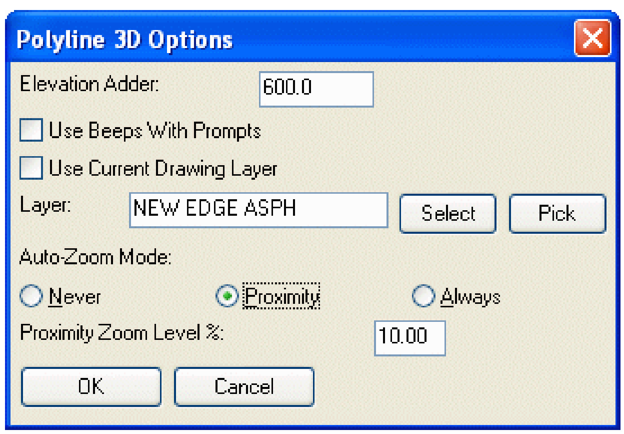


At the command line, enter in the building elevation of 634.41 found labeled in the middle of the building and press Enter. Then pick the points that define the building outline. Start in the upper left corner and pick at every corner around the building. When you have picked around the entire building, type in [A] for close to finish digitizing the building.

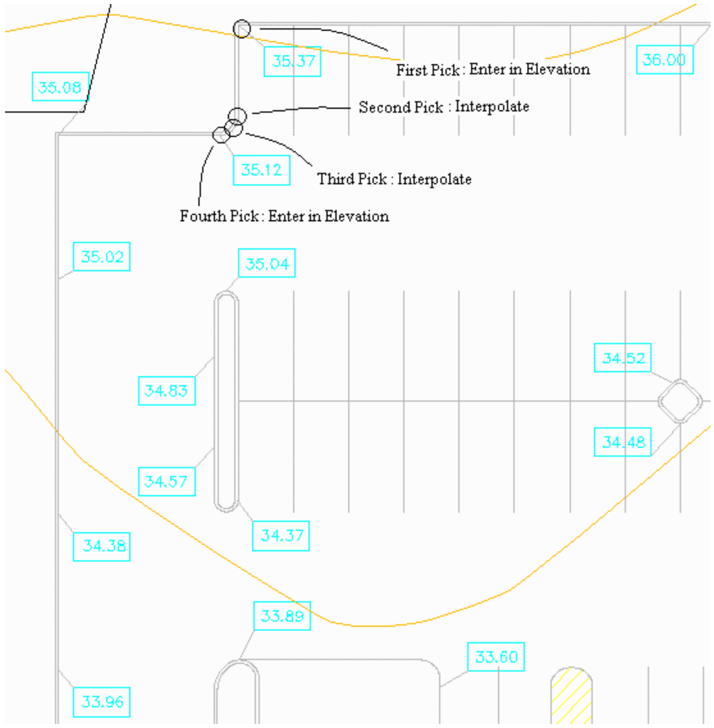
Enter polyline elevation <0.00>: *634.41*
First point: *pick a building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick the next building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick the next building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick the last building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *A to close*

Digitize Another NEW_BUILD Polyline [Yes(A)/<No(B)>]? *B for No*

Notice that the parking lot linework consists of different elevation levels. To digitize entities with more than one elevation, go to Digitize and select 3D Polyline from the pull-down menu. Make sure that the **Prompt For Polyline Elevation** option is on, the **Use current drawing layer** toggle is off and name the layer NEW EDGE ASPH.



Let's start by digitizing the parking lot starting from the zoomed in section below. The edge of asphalt is the inside line. The parking lot elevation labels have been shortened on the paper plan. For example, they read 35.37 and 35.12, when the actual elevations are 635.37 and 635.12. Enter in 600 as the Elevation Adder, then click OK.



Click on the point with the digitizer puck where the 35.37 elevation label points to in the upper left corner of the parking lot. When prompted for Elevation enter in 35.37. Pick below the first point where the linework starts to curve. We do not have an elevation for this point, but we can interpolate the elevation from the two points around it using the interpolate option. Type in I for interpolate or hit the A button on the Puck. Next pick the middle point of the curve and again use Interpolate for the elevation. Next pick the end of the curve at the 35.12 label and enter in the elevation 35.12. Continue digitizing for the rest of the edge of asphalt linework. Digitize each point where there is an elevation label and each point where the curb line changes direction.

The first prompts should resemble these:

First point: *pick first point (at 35.37 label)*
Interpolate[A]/screen Pick/<Elevation[B]> <0.00>: *35.37*

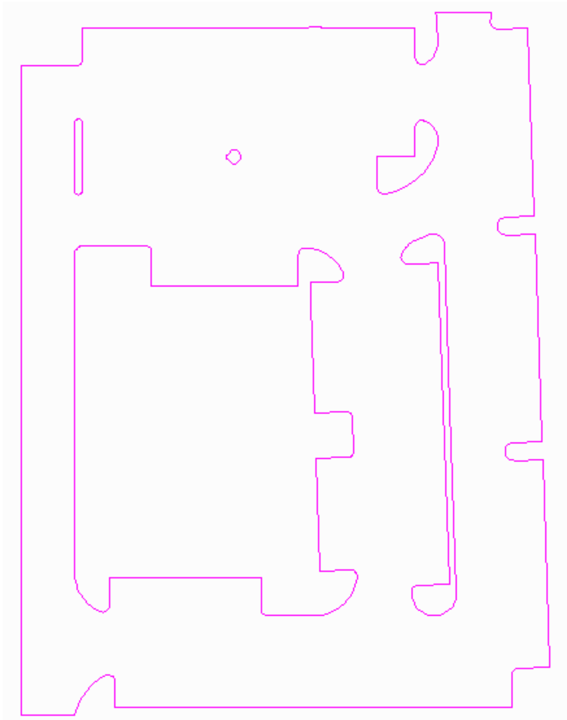
Z: 635.37
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick next point (start of curve)*
Slope/Ratio/Interpolate[A]/Degree/screen Pick/<Elevation[B]> <635.37>: Press the [A] button on the Puck for Interpolate

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: *pick next point (middle of curve)*
This point elevation will be interpolated upon completion.
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: *pick next point (end of curve, at 35.12 label)*
This point elevation will be interpolated upon completion.

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: *35.12 (Enter)*

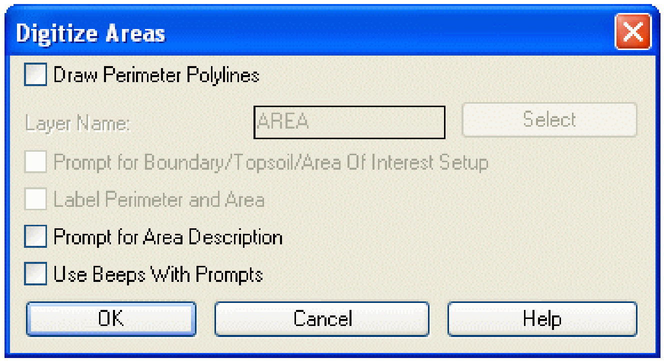
To check the elevations of the interpolated points go to List under the Inquiry menu and click on the polyline you just created and press Enter. A text window will appear showing you the layer name, coordinates, and elevation of each point. To return to the main graphic screen, press F2.

Use the 3D Polyline command to digitize the rest of the parking lot as seen below.



Step 5 (Area):

Now that we have digitized the Design Surface, let's check the Area of certain sections. Select Area under the Digitize Menu and match the below dialog.



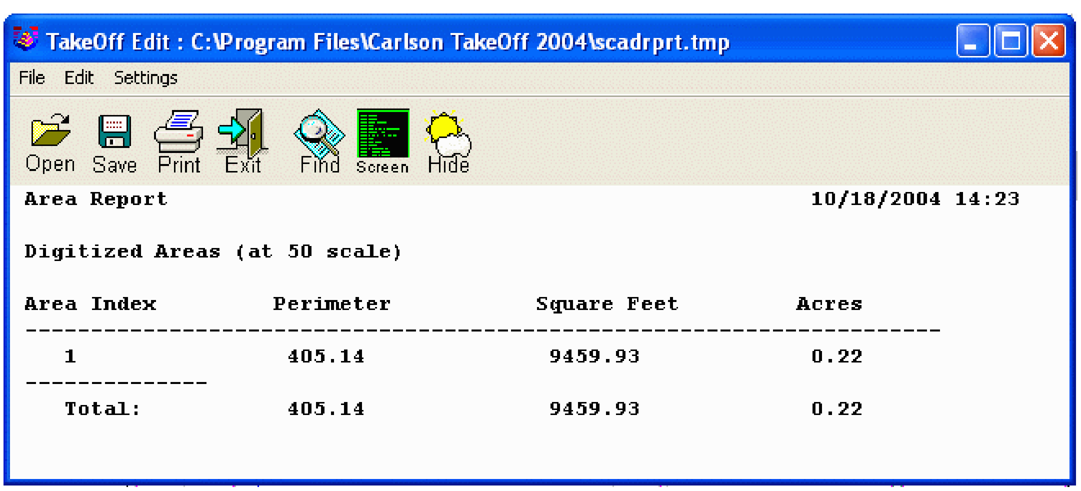
To approximate the area of the main building, pick the points of the building outline.

Command: *dig_area*

Pick starting point: *Pick points as close to the building design linework as you can*
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):

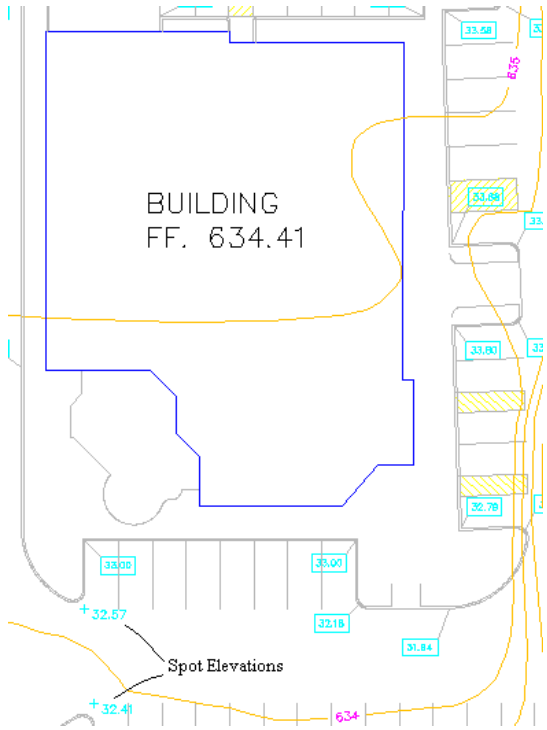
Digitize Another Area [<Yes(A)>/No(B)]? *B*

When finished with the building points, press Enter to end. Then answer no for no more areas. Takeoff will then display an Area report similar to the one shown below.

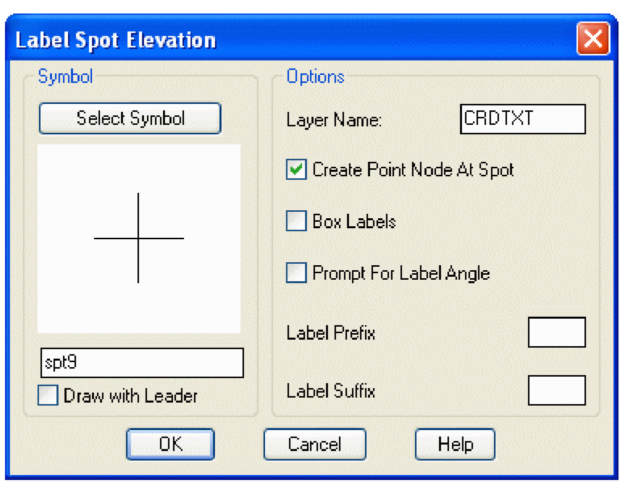


Step 6 (Spot Elevations):

In our paper drawing, we have two spot elevations labeled 32.57 and 32.41 shown in the bottom left below.



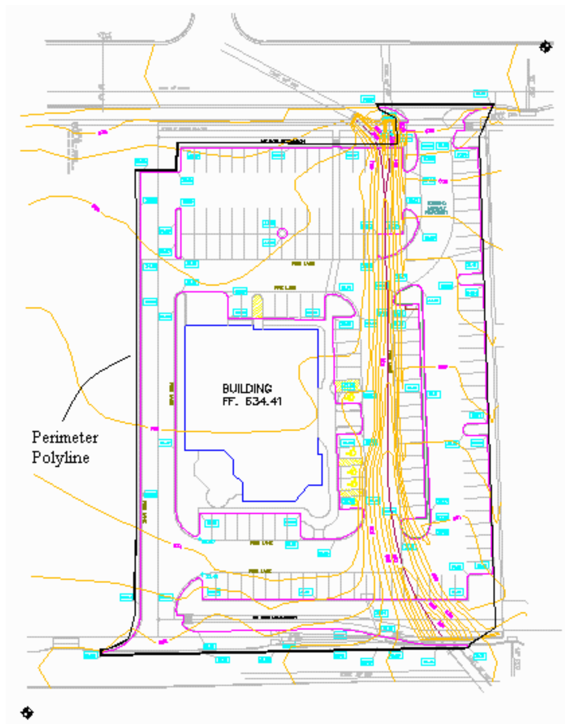
To digitize these elevations, we can use the Spot Elevation command under the Digitize menu. Fill out the Spot Elevation dialog as shown and pick OK.



In the paper plan, find and click on the spot elevations with the puck. When prompted, enter in their corresponding elevations of 632.57 and 632.41.

Step 7 (Boundary Polyline):

The limits of the site are defined by a closed polyline. This polyline is used as the boundary for the models and the volumes. Under the digitize menu, check on Other and then select Perimeter. Type in PERIMETER as the layer name. Now digitize around the bold, outside line shown below.



Say No to the prompt: **Digitize Another PERIMETER Polyline [Yes(A)/<No(B)>]?**

Now run Tools->Boundary Polyline->Set Boundary Polyline and pick the perimeter polyline. This selected polyline is now set as the boundary polyline for the rest of the Takeoff routines.

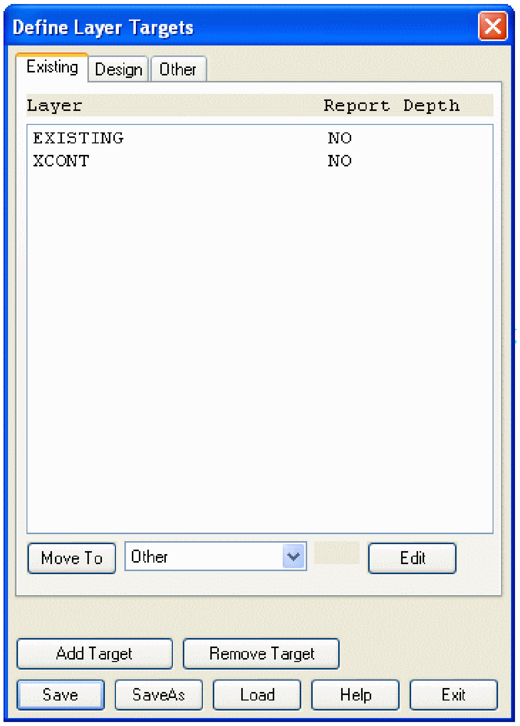
Step 8 (Layer Targets):

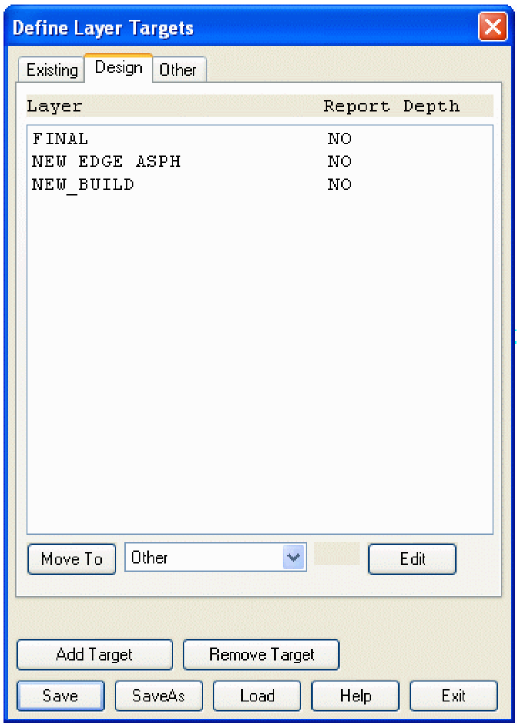
From the Tools menu, choose Define Layer Target/Material/Subgrade. Every entity (line, polyline, point, etc) in the drawing is assigned a layer name. Takeoff uses the entity layer names to define which entities are for the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces. The drawing entities are assigned their target surface by their layer name. For example, if polylines representing design contours are on the layer "NEW", then "NEW" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other.

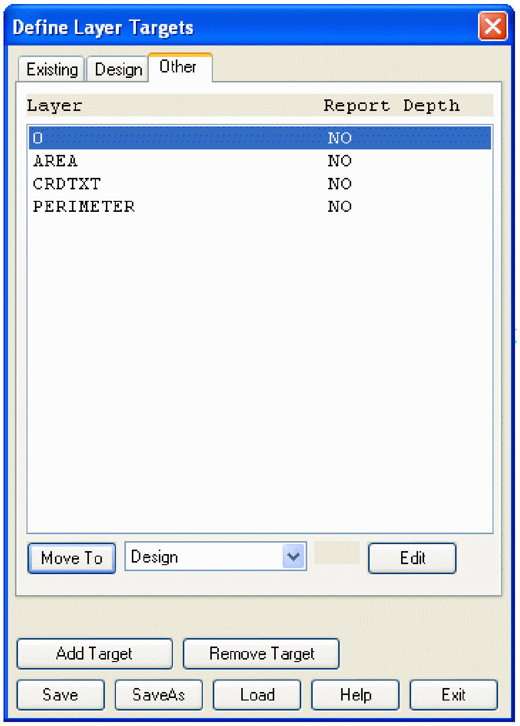
The Define Layer Targets dialog has three lists of layers: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog. We have already defined the layers for their correct targets. We did this by check on

Existing, Design, or Other in the pull-down menu.

Check that your Layer Targets resemble the three lists shown here. If a layer is out of place, highlight it, and hit the "Move To" button after selecting the correct target to send it to. After reviewing, pick Save and Exit.







Now that the layer targets are defined, there are several commands that can be applied. In the Display menu, you can turn on/off whether to display layer targets by using Existing Drawing, Design Drawing and Other Drawing, or by right-clicking with your mouse. For example, when Design Drawing is checked, then picking this menu item will uncheck it and turn off all the layers for the design surface. Likewise, picking Design Drawing when it is unchecked will make it checked and turn on the design surface layers.

Practice turning on/off the Existing, Design and Other Drawing in the Display menu. When only Existing Drawing is on, you should see just the contours. When only Design Drawing is on, you should see just the design polylines and leader labels. When only Other Drawing is on, you should see the entities that are assigned to neither existing nor design.

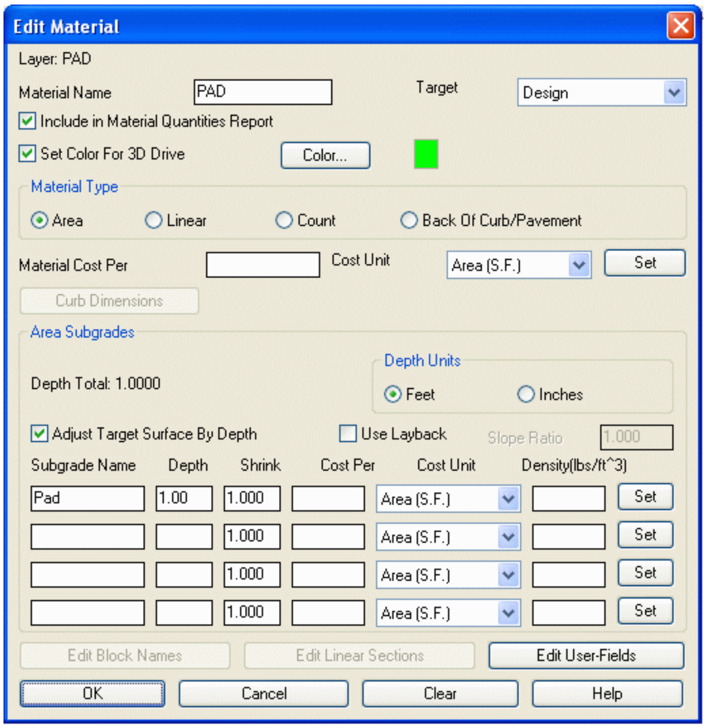
Step 9 (Define Material/SubGrade):

Besides assigning target surfaces by layer, layers are also used to define material names and subgrade depths. By assigning material names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface. The polylines used for subgrade depth must be closed polylines. Takeoff supports nested subgrade polylines for exclusion areas such as islands by counting how

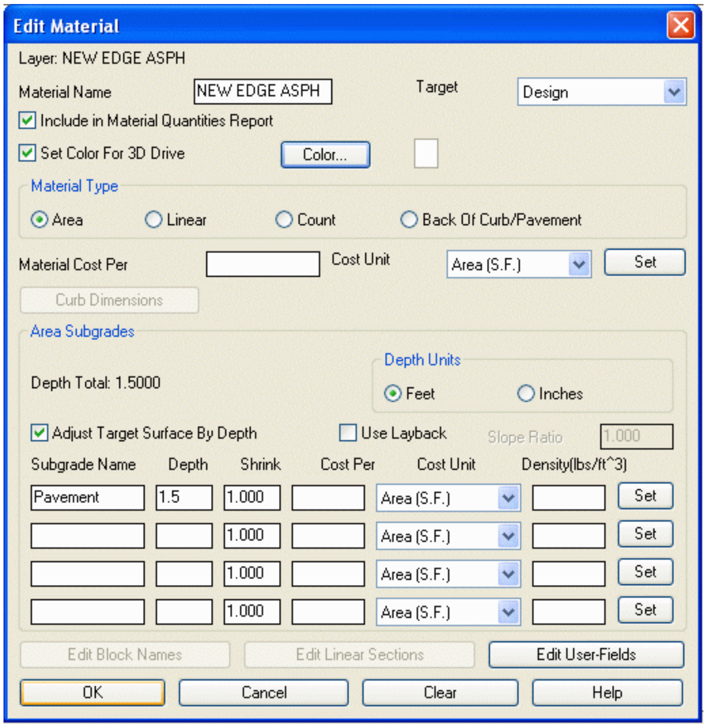
many subgrade polylines surround an area. If the number is odd, then the area is inside the subgrade. Otherwise the area is not part of the subgrade.

First, let's confirm the layer names for our subgrades. Go to the Display menu and check on Design Drawing, uncheck Existing Drawing and uncheck Other Drawing. Then run Inquiry->Layer ID and pick the large pad polyline. It reports that this layer is NEW BUILD. Next use Layer ID to pick the curb polyline. It reports that this layer is NEW EDGE ASPH.

Now run Define Layer Target/Material/Subgrade and pick the Design tab. Highlight layer NEW BUILD and pick the Edit button. A dialog appears for defining the pad material properties. Check on the Include In Material Report option, enter the Material Name as "Pad", set the first subgrade name to "Pad", and set the Depth as 1. Once the dialog is filled out as shown, pick OK.

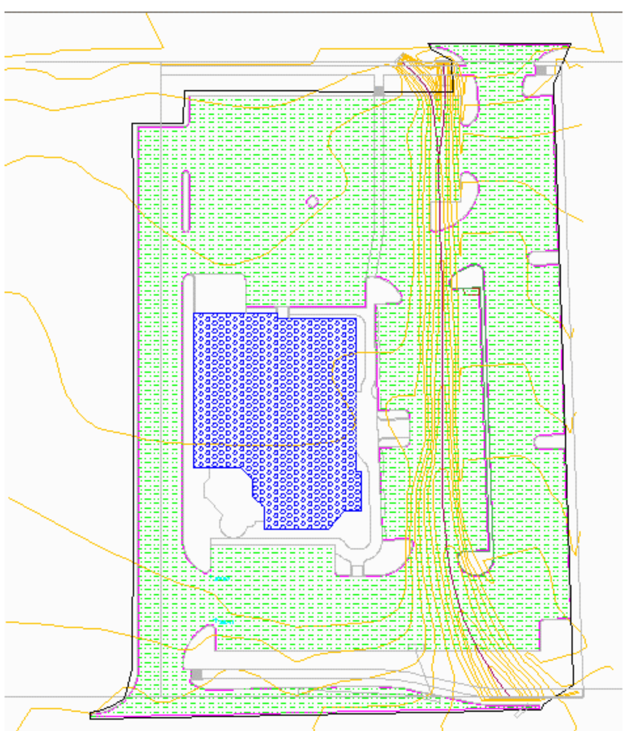


Next pick layer NEW EDGE ASPH and choose Edit. In the Edit Materials dialog, check on Include In Material Report, set the Material Name to "Pavement", set the first subgrade name to "Pavement", and set the Depth to 1.5. Then pick OK.



To save the subgrade changes, pick the Save button on the Define Layer Targets dialog. Then choose Exit.

Now let's visually verify the subgrade areas. In the TakeOff menu, run Subgrade Areas->Hatch Subgrade Areas. There is a dialog to select which subgrade to hatch. Choose the Pavement. Then there is a dialog for the hatch pattern and color. Change the color to green and click OK. Then run Hatch Subgrade Areas again. This time choose Pad and set the hatch pattern to Hex with blue color. The resulting hatch areas show where the subgrade is applied. Notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. When finished viewing the subgrade areas, run TakeOff->Subgrade Areas->Erase Subgrade Hatches.

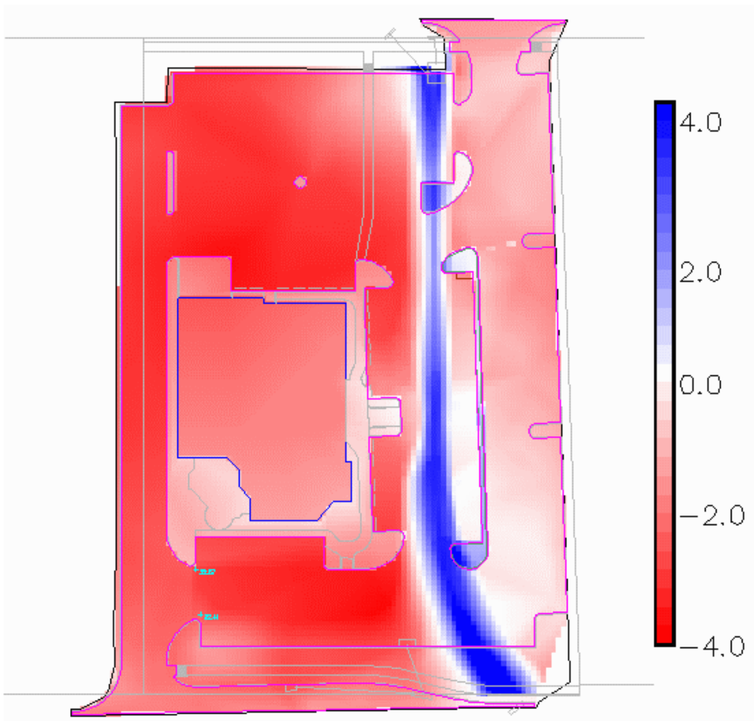


Step 10 (Model Existing and Design Surfaces):

To calculate volumes, Takeoff needs two surfaces: existing ground and design. These surfaces are modeled by triangulation. With the preparation of the previous steps, we're now ready to make the models. To make the existing ground surface, run Tools->Make Existing Ground Surface. The program will process the entities and make the triangulation surface. Then to make the design surface, run Tools->Make Design Surface.

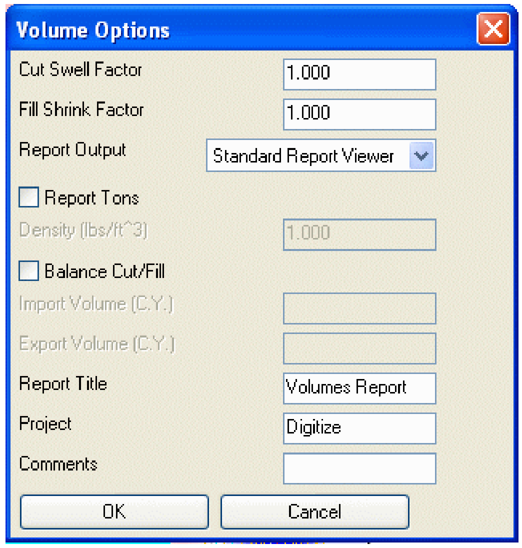
Step 11 (Cut/Fill Color Map):

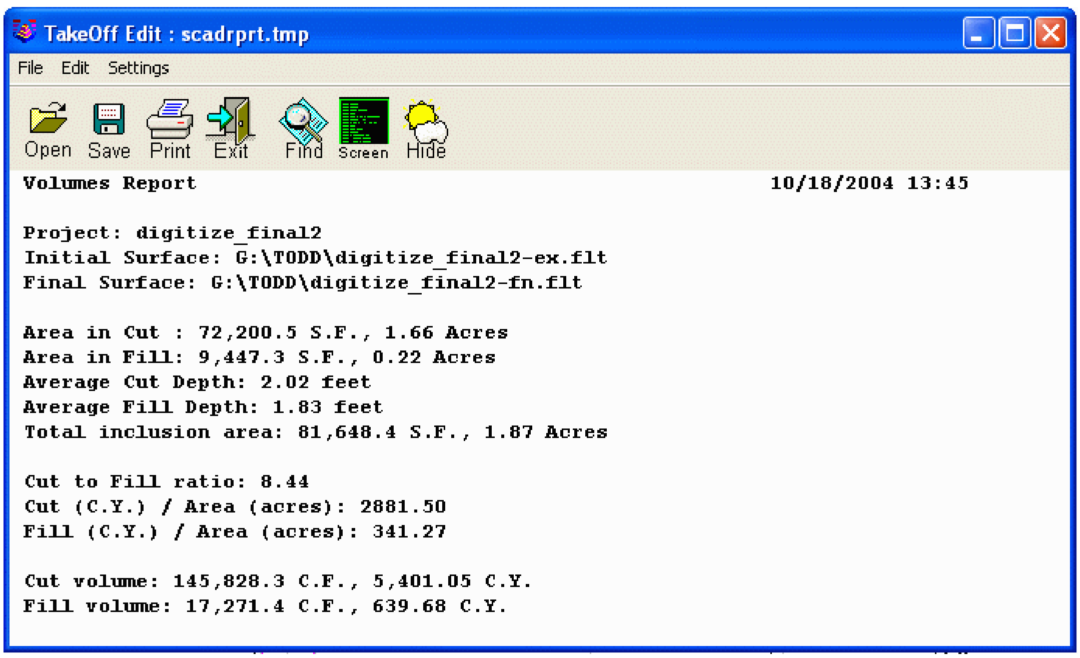
Cut/Fill color maps can be used for a visual output of the site cut/fill areas and also serves as a check that the models are correct. In the Display menu, choose Cut/Fill Color Map. Cut areas are drawn in different shades of red for different depths of cut while fill areas are drawn in blue. To change the resolution of the color blocks, run Display->Display Options and change the Cut/Fill Color Map Subdivisions. This parameter is the number of rows and columns of color blocks to create. You can also draw a legend for the Color Map by going to Draw, Cut/Fill Map Legend. Pick a point on your drawing to locate the legend and press Enter. To turn off the color map, go to the Display menu and pick Cut/Fill Color Map to uncheck it.



Step 12 (Calculate Volumes):

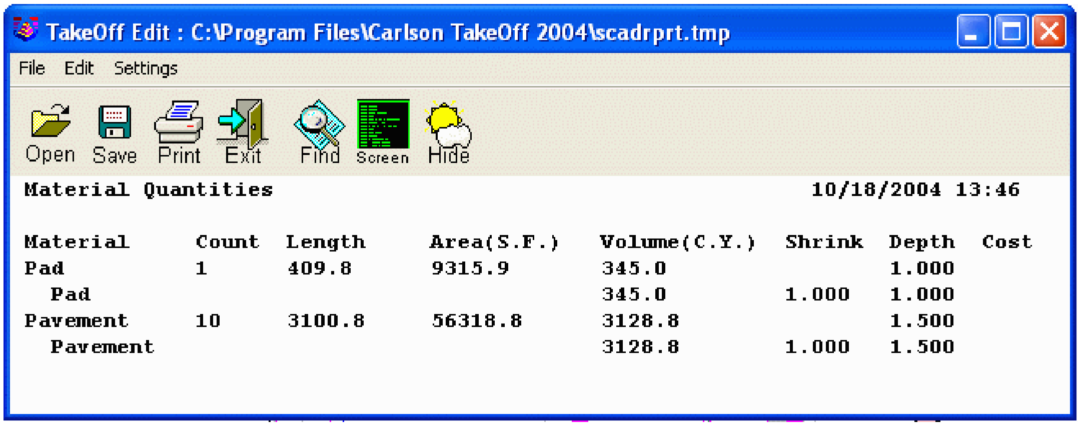
To calculate volumes, run the TakeOff->Calculate Total Volumes command. There is an options dialog for setting the cut swell factor and fill shrink factor. These values get multiplied into the cut/fill volumes. Set these factors as desired and click OK. Then the routine calculates the volumes and display the report which includes the cut/fill volumes and areas. The report can be printed or saved to a file. Pick the Exit button to exit the report viewer.





Step 13 (Material Quantities):

To report the material quantities, run the TakeOff->Material Quantities->Standard Report routine. The report includes the count, length, area and volume for each type of material that was assigned for reporting in the Define Layer Target/Material/Subgrade command. The Material Quantities->Custom Report routine can be used to reporting these values with control of the report format and the option to export to Excel.


TakeOff Edit : C:\Program Files\Carlson TakeOff 2004\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Material Quantities 10/18/2004 13:46
Material Count Length Area(S.F.) Volume(C.Y.) Shrink Depth Cost
Pad 1 409.8 9315.9 345.0 1.000
Pad 345.0 1.000 1.000
Pavement 10 3100.8 56318.8 3128.8 1.500
Pavement 3128.8 1.000 1.500

Index